

LEGAL NOTICE

This report was prepared by Ontario Hydro and, while effort has been made to assure the accuracy and applicability of the material contained herein, neither Ontario Hydro nor any person acting on its behalf:

- (a) makes any warranty, express or implied with respect to the use of any information, apparatus, method or process disclosed in this report or that such use may not infringe privately owned rights; or
- (b) assumes any liabilities with respect to the use of, or for damages resulting from the use of, any information, apparatus, method or process disclosed in this report.



**ontario hydro
research division**

COMPLIMENTARY COPY
Please Credit Ontario Hydro Research if
Reproduced in Whole or in Part

H3DMAP VERSION 4 A GENERAL THREE-DIMENSIONAL FINITE ELEMENT COMPUTER CODE FOR LINEAR AND NON-LINEAR ANALYSES OF STRUCTURES

USER MANUAL AND VERIFICATION CASES - VOLUME 1, REVISION 1

Report No. 89-221-K

R.G. Sauvé, Unit Head-Applied Mechanics Development

E. Nadeau, Engineer

Applied Mechanics Section

Mechanical Research Department

ABSTRACT

The finite element computer code H3DMAP Version 4 is a general three dimensional structural analysis package that has been developed at Ontario Hydro to solve a wide variety of problems encountered by the corporation. The code has been extensively benchmarked and verified for a number of cases using both experimental results and classical solutions. The program is based on a highly modular architecture and includes the following simulation modules: static analysis for thermal and mechanical loads, modal analysis, frequency response, response spectrum generation, flow induced vibration, seismic analysis (response spectrum method), buckling/stability analysis, linear and non-linear geometric transient dynamic analyses using the implicit method, geometric and material non-linear explicit transient dynamic analysis using the hydrodynamic formulation, shock and wave propagation including interaction problems.

... (cont'd)

job	21-562-051	file	825.201	date	April 15, 1991	report no.	89-221-K
-----	------------	------	---------	------	----------------	------------	----------

H3DMAP includes the following types of elements: straight and curved Euler-Bernoulli (cubic) beams, isoparametric thin/thick beams or trusses, pipes, isoparametric thin/thick shells, isoparametric 2-D/axisymmetric and 3-D continuum, spring elements and analog elements. In addition, the program handles a variety of material models: elastic, elastic-plastic, hydrodynamic (e.g. fluid) with equation of state and isotropic-hydrodynamic elastic-plastic with equation of state. The non-linear models are valid for finite deformation (i.e. severe straining and rotation). A three dimensional contact-impact algorithm is included in the explicit module for modelling sliding material interfaces. This option along with that of the specified velocity boundary condition, makes the code well suited to both crashworthiness and manufacturing problems. A dynamic relaxation solution procedure permits severely deforming bodies under relatively static loading to be handled effectively.

H3DMAP is augmented by comprehensive post-processing and graphics capabilities. The program is written in FORTRAN 77 and is currently operational on DEC computers and workstations under both VAX/VMS and ULTRIX operating systems, Silicon Graphics and APOLLO computers under UNIX, and 386 based PC's under DOS. In addition, it can be easily ported to other environments. The report consists of two volumes: Volume 1 describes the verification cases used to validate the computer code, and Volume 2 contains the user manual and sample input data corresponding to the test cases in Volume 1.



ontario hydro research division

EXECUTIVE SUMMARY

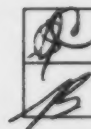
H3DMAP VERSION 4 A GENERAL THREE-DIMENSIONAL FINITE ELEMENT COMPUTER CODE FOR LINEAR AND NON-LINEAR ANALYSES OF STRUCTURES

USER MANUAL AND VERIFICATION CASES - VOLUME 1, REVISION 1

The finite element computer code H3DMAP Version 4 is a general three dimensional structural analysis package that has been developed at Ontario Hydro to solve a wide variety of problems encountered by the corporation. H3DMAP is an acronym for **Hydro 3 Dimensional Matrix Analysis Program**. The code has been extensively benchmarked and verified for a number of cases using both experimental results and classical solutions. The program is based on a highly modular architecture with simulation modules capable of handling the following procedures:

- static analysis of thermal and mechanical loads
- modal analysis
- frequency response
- response spectrum generation
- flow induced vibration
- seismic analysis (response spectrum method)
- buckling/stability analysis
- linear and non-linear geometric transient dynamic analyses using the implicit method
- geometric and material non-linear explicit transient dynamic analysis using the hydrodynamic formulation
- shock and wave propagation including interaction with hydrodynamic materials.

H3DMAP includes the following types of elements: straight and curved Euler-Bernoulli (cubic) beams, isoparametric thin/thick beams or trusses, pipes, isoparametric thin/thick shells, isoparametric 2-D/axisymmetric continuum, isoparametric 3-D continuum, spring elements and analog elements. In addition, the program handles a variety of material models: elastic, elastic-plastic, hydrodynamic (e.g. fluid) with equation of state and isotropic-hydrodynamic elastic-plastic with equation of state.



cont'd ...

job	21-562-051	file	825.201	date	April 15, 1991	report no.	89-221-K
-----	------------	------	---------	------	----------------	------------	----------

Among the salient features of H3DMAP are the flow induced vibration option and the non-linear explicit transient dynamic option for finite deformation. A three-dimensional contact-impact algorithm using the concept of master-slave surfaces is included in the explicit module for modelling sliding material interfaces. This option along with that of specified velocity boundaries makes the code well suited to both crashworthiness and manufacturing problems. A dynamic relaxation procedure permits non-linear statically loaded bodies to be handled effectively.

H3DMAP is augmented by comprehensive post-processing and graphics capabilities. The program is written in FORTRAN 77 and is currently operational on DEC computers and workstations under both VAX/VMS and ULTRIX operating systems, Silicon Graphics and APOLLO computers under the UNIX operating systems, and 386 based PC's under DOS. With minimum effort, the program can be ported to other environments. Because of its modular structure, the code can also be modified to suit particular applications with minimum effort. The report consists of two volumes: Volume 1 describes the verification cases used to validate the computer code, and Volume 2 contains the user manual and sample input data corresponding to the test cases in Volume 1.

TABLE OF CONTENTS

	<u>Page</u>
1.0 INTRODUCTION	1
2.0 VERIFICATION CASES	3
2.1 Case 1 - Straight Beam Element	3
2.2 Case 2 - Curved Beam Element	3
2.3 Case 3 - Member Degree of Freedom Releases	3
2.4 Case 4 - Piping Test Loop Eigensolution	4
2.5 Case 5 - Seismic Analysis	4
2.6 Case 6 - Flow Induced Vibration	5
2.7 Case 7 - Linear Transient Response	5
2.8 Case 8.1, 8.2 - Initial Velocity Response	6
2.9 Case 9 - Non-Linear Transient Dynamics (Boundary Element)	6
2.10 Case 10 - Non-Linear Transient Dynamics (Inter-Element)	6
2.11 Case 11 - Non-Linear Transient Dynamics (Annular Element)	7
2.12 Case 12 - Fretting Wear Force Prediction (Benchmark with AECL)	7
2.13 Case 13 - Sine Beat Verification	7
2.14.1 Case 14.1 - Simple Frame Stability Analysis	8
2.14.2 Case 14.2 - Euler Column Buckling Analysis	8
2.15 Case 15 - Lateral Stability of a Cantilever Beam	8
2.16.1 Case 16.1 - Straight Beam Element - Thermal Growth Test	8
2.16.2 Case 16.2 - Straight Beam Element - Thermal Load Test	9
2.17 Case 17 - Frame Structure - Mixed Elements	9
2.18 Case 18 - Curved Beam Element - Thermal Load Test	9
2.19 Case 19 - Fretting Wear Force Prediction: Verification with Experiment	9
2.20 Case 20 - Pipe Whip Analysis: Verification with Experiment	9
2.21 Case 21 - Single Degree of Freedom Model - Displacement Response	10
2.22 Case 22 - Dynamic Relaxation Effects	10
2.23 Case 23 - Thin Cylindrical Shell Roof Analysis	12
2.24 Case 24 - Cylinder Under Internal Pressure	13
2.25 Case 25 - Simply Supported Square Plate Analysis	13
2.25.1 Case 25.1 - Concentrated Load at Plate Centre	13
2.25.2 Case 25.2 - Uniform Pressure Load	13
2.25.3 Case 25.3 - Dynamic Modal Analysis	13
2.25.4 Case 25.4 - Clamped Plate Under Thermal Gradient	13
2.25.5 Case 25.5 - In-Plane Expansion	14
2.26 Case 26 - Clamped Hyperbolic Paraboloid Shell	14
2.27 Case 27 - Linear Dynamic Response of a Spherical Shell	14
2.28 Case 28 - Pinched Circular Cylinder Test Case	14
2.29 Case 29 - Stability of a Simply Supported Square Plate Uniformly Compressed in One Direction	15
2.30 Case 30 - Euler Shell Buckling Analysis	15
2.31 Case 31 - Calandria Tube Shell Buckling Analysis	15
2.32 Case 32 - Corner Supported Plate	16
2.33 Case 33 - Isoparametric Beam Element Test	16
2.33.1 Case 33.1 - Concentrated Loads at Centre	16

Table of Contents ... 2

		<u>Page</u>
2.33.2	Case 33.2 - Thermal Expansion	16
2.33.3	Case 33.3 - Dynamic Modal Analysis	16
2.33.4	Case 33.4 - Isoparametric Beam Column Buckling	17
2.34	Case 34 - Cantilever Beam	17
2.35	Case 35 - Thin-Wall Vessel	18
2.36	Case 36 - Thick-Wall Vessel	18
2.37	Case 37 - Stress Concentration - Hole in a Flat Plate	19
2.38	Case 38 - Simply Supported Circular Disk - Natural Frequencies	19
2.39	Case 39 - Simply Supported Beam - Natural Frequencies	19
2.40	Case 40 - Axisymmetric Buckling of a Circular Simply-Supported Plate	20
2.41	Case 41 - Impulsively Loaded Beam	20
2.42	Case 42 - Axial Impulsively-Loaded Beam	21
2.43	Case 43 - Geometric Non-Linear Beam Test	21
2.44	Case 44 - Moment Load Test for Non-Linear Beam (Dynamic Relaxation)	21
2.45	Linear and Geometric Non-Linear Shell Test	22
2.46	Case 46 - Simply Supported Square Plate Test (Elastic and Elastic-Plastic Cases)	22
2.47	Case 47 - Impulsively Loaded Cylindrical Panel	23
2.48	Case 48 - Impulse Load on a Circular Plate	23
2.49	Case 49 - Moment Load Test for Non-Linear Shell (Dynamic Relaxation)	23
2.50	Case 50 - Corner Supported Flat Plate Under Pressure Load (Explicit Solution)	24
2.51	Case 51 - Axial Impact of a 3-D Beam - Rigid Wall	24
2.52	Case 52 - Axial Impact of a 3-D Beam - Boundary Restrictions	24
2.53	Case 53 - Simply Supported Beam - Step Load at Mid-Span	24
2.54	Case 54 - Axial End Load on a Cantilever Beam	25
2.55	Case 55 - Dynamic Relaxation	25
2.56	Case 56 - Geometric Non-Linear Beam Test	25
2.57	Case 57 - Undamped Spring-Mass System with Initial Velocity	25
2.58	Case 58 - Undamped Spring-Mass System with Force Step Function	25
2.59	Case 59 - Undamped Spring-Mass System - Beat Response	25
2.60	Case 60 - Annular Element	26
2.61	Case 61 - Sandwich Model - Dynamic Relaxation	26
2.62	Case 62 - Fluid-Solid Interaction of Tritiated Heavy Water Transportation Container	26
2.63	Case 63 - Fluid Hydrodynamic Material Model Testing Equations of State Type 1 and 2	27
2.64	Case 64 - Dynamic Relaxation Test - 3-D Cantilever Beam	28
2.65	Case 65 - Pipe Whip Test	28
2.66	Case 66 - Tritiated D ₂ O Transport Container Punch Test	28
2.67	Case 67 - Corner Drop Model	30
2.68	Case 68 - Sphere Impact Experiment	30
2.69	Case 69 - Radioactive Filter Transportation Package 9m Corner Drop Impact	30
2.70	Case 70 - Rolled Joint Simulation	32
2.71	Case 71 - Contact Algorithm Tests	34

Table of Contents ... 3

ACKNOWLEDGMENTS	35
REFERENCES	36
DISTRIBUTION	last
3.0 H3DMAP VERSION 4 USER MANUAL	Vol. 2
ACKNOWLEDGMENTS	54
REFERENCES	55
DISTRIBUTION	last

LIST OF TABLES

Table 1	Shear Area Coefficient K	Vol. 2
Table 2	Damping Values	Vol. 2
Table 3	Formulae for Calculating Beta and Gamma for Rayleigh Proportional Damping	Vol. 2

FIGURES

APPENDIX 1 - VMS and UNIX Command Procedures	Vol. 2
APPENDIX 2 - H3DMAP Input Files	Vol. 2

LIST OF FIGURES

- 1.1 H3DMAP Version 4 Overall Program Structure Modules
- 1.2 Node Numbering Optimization Routines
- 1.3 Assembly, Element Routines
- 1.4 Cubic and Isoparametric Beam Element Routines
- 1.5 Shell Element Routines
- 1.6 Load, Mass Routines
- Inverse Iteration Routines
- Subspace Iteration Routines
- 1.7 Implicit Dynamic Transient Analysis Routines
- 1.8 Frequency Response Analysis Routines
- Flow Induced Vibration Analysis Routines
- Seismic Response Analysis Routines (Response Spectrum Method)
- 1.9 Linear Static Analysis Routines
- Structural Stability Routines
- 1.10 Linear Stress Recovery Routines
- 1.11 Explicit Transient Dynamic Analysis Routines
- 1.12 Explicit Beam Element Routines
- Explicit Shell Element Routines
- Explicit Analog Element Routines
- 1.13 Explicit Three-Dimensional Continuum Element Routines
- 1.14 Overall Computer Code Capabilities

- 2.1.1 Case 1 - Computer Model (Straight Beam Test)
- 2.2.1 Case 2 - Computer Model (Curved Beam Test)
- 2.3.1 Case 3 - Computer Model (Member Release Test)
- 2.4.1 Case 4 - Piping Test Loop Computer Model
- 2.4.2 Case 4 - Piping Test Loop Node Numbers
- 2.4.3 Case 4 - Piping Test Loop Boundary Conditions
- 2.4.4 Case 4 - Piping Test Loop Mode Shape 1
- 2.4.5 Case 4 - Piping Test Loop Mode Shape 2
- 2.4.6 Case 4 - Piping Test Loop Mode Shape 3
- 2.4.7 Case 4 - Piping Test Loop Mode Shape 5
- 2.5.1 Case 5 - Computer Model (Eigensolution, Seismic Test)
- 2.6.1 Case 6 - AECL Chalk River Flow Induced Vibration Data
- 2.6.2 Case 6 - Flow Induced Vibration Computer Benchmark Model
- 2.7.1 Case 7 - Linear Transient Displacement Response vs. Time
- 2.8.1 Case 8.1 - H3DMAP Comparison with Analytical Results for Initial Velocity ($K = .60 \times 10^8 \text{ lb/in}$)
- 2.8.2 Case 8.2 - H3DMAP Comparison with Analytical Results for Initial Velocity ($K = .60 \times 10^6 \text{ lb/in}$)
- 2.9.1 Case 9 - Non-Linear Transient Dynamic Displacement Response
- 2.10.1 Case 10 - Velocity Response During Momentum Transfer
- 2.11.1 Case 11 - Non-Linear Annular Element Test Model
- 2.11.2 Case 11 - Orbit of Node 4 in an Annulus
- 2.12.1 Case 12 - Fretting Force Prediction Benchmark Model
- 2.12.2 Case 12 - Displacement Response vs. Time at Node 25
- 2.12.3 Case 12 - Impact Force vs. Time at Node 25
- 2.12.4 Case 12 - Velocity Response vs. Time at Node 25
- 2.12.5 Case 12 - Force vs. Time at Support No. 4 (Node 25) CRNL Model
- 2.13.1 Case 13 - Sine Beat Response - Displacement vs. Time
- 2.14.1 Case 14.1 - Frame Stability Analysis - Computer Model

List of Figures ... 2

2.14.2	Case 14.2 - Euler Column Buckling - Computer Model
2.15.1	Case 15 - Lateral Buckling Analysis - Computer Model
2.16.1	Case 16.1 - Straight Beam Thermal Growth - Computer Model
2.16.2	Case 16.2 - Straight Beam Thermal Load - Computer Model
2.17.1	Case 17 - Mixed Frame Test - Computer Model
2.18.1	Case 18 - Curved Beam Thermal Growth - Computer Model
2.19.1	Case 19 - Experimental Benchmark Model
2.19.2	Case 19 - Measured/Predicted Displacement and Impact Force Response vs. Time at Node 9
2.20.1	Case 20 - Pipe Whip Benchmark Model
2.21.1	Case 21 - Single Degree of Freedom Model - Displacement Response
2.22.1	Case 22 - Dynamic Relaxation Computer Model and Static Response
2.22.2	Case 22 - Dynamic Relaxation Displacement vs. Time Curve at Node 2
2.23.1	Case 23 - Thin Cylindrical Shell Roof - Computer Model
2.23.2	Case 23 - Thin Cylindrical Shell Roof - Longitudinal and Vertical Static Displacements
2.23.3	Case 23 - Thin Cylindrical Shell Roof - Bending/Membrane Stresses
2.24.1	Case 24 - Thin Cylinder Under Internal Pressure - Computer Model
2.25.1	Case 25 - Simply Supported Square Plate Analysis - Computer Model
2.25.2	Case 25 - Simply Supported Square Plate Modal Analysis - Mode Shape 1
2.25.3	Case 25 - Simply Supported Square Plate Modal Analysis - Mode Shape 2
2.25.4	Case 25 - Simply Supported Square Plate Modal Analysis - Mode Shape 4
2.25.5	Case 25 - Simply Supported Square Plate Modal Analysis - Mode Shape 5
2.26.1	Case 26 - Clamped Hyperbolic Paraboloid Shell - Computer Model
2.26.2	Case 26 - Deflection at Centre Line of Hyperbolic Paraboloid Shell
2.27.1	Case 27 - Spherical Shell Dynamic Response - Computer Model
2.27.2	Case 27 - Spherical Shell Dynamic Response at Node 1
2.28.1	Case 28 - Pinched Circular Cylinder - Computer Model
2.29.1	Case 29 - Square Plate Stability Analysis - Computer Model
2.29.2	Case 29 - Square Plate Stability Analysis - Mode Shape 1
2.30.1	Case 30 - Euler Shell Buckling Analysis - Computer Model
2.30.2	Case 30 - Euler Shell Buckling Analysis - Model Shape 1
2.31.1	Case 31 - Calandria Tube Shell Buckling Analysis - Computer Model
2.31.2	Case 31 - Calandria Tube Shell Buckling Analysis - Mode Shape 1
2.32.1	Case 32 - Corner Supported Plate Test
2.33.1	Case 33 - Isoparametric Beam Element Test
2.34.1	Case 34 - 2-D Cantilever Beam Models
2.34.2	Case 34 - 3-D Cantilever Beam Models
2.35.1	Case 35 - Thin Wall Vessel Model
2.36.1	Case 36 - Thick Wall Cylinder Model
2.37.1	Case 37 - Stress Concentration Evaluation - Computer Model
2.38.1	Case 38 - Simply Supported Circular Disk - First Three Mode Shapes
2.39.1	Case 39 - Simply Supported Beam - First Three Mode Shapes
2.40.1	Case 40 - Axisymmetric Buckling of a Circular Plate Geometry, Model, Load and 1st Mode Shape
2.41.1	Case 41 - Impulsively Loaded Beam Model
2.41.2	Case 41 - Impulsively Loaded Beam - Deflection at Node 11
2.42.1	Case 42 - Impulsively Axially Loaded Beam
2.43.1	Case 43 - Geometric Non-Linear Beam Test - Model and Tip Deflection Response
2.43.2	Case 43 - Geometric Non-Linear Beam Test - Response at Time: 0.0027 sec.
2.44.1	Case 44 - Moment Load Test for Non-Linear Beam - Model and Displacement at Node 13
2.44.2	Case 44 - Moment Load Test for Non-Linear Beam - Response at Time: 0.0010 sec.

List of Figures ... 3

2.44.3	Case 44 - Moment Load Test for Non-Linear Beam - Response at Time: 0.0050 sec.
2.45.1	Case 45 - Geometric Non-Linear Cantilever Shell Test - $P=2.85$ psi- Model and Displacement Response of Node 6 and 12 in the Y-Direction
2.45.2	Case 45 - Geometric Non-Linear Cantilever Shell Test - $P=2.85$ psi- Response at Time = 4ms
2.46.1	Case 46 - Simply Supported Square Plate - Test Model, Material Properties and Applied Load
2.46.2	Case 46 - Displacement at Node 25 in the Y-Direction - Elastic- Plastic Material
2.46.3	Case 46 - Displacement at Node 25 in the Y-Direction - Elastic Material
2.46.4	Case 46 - Response at Time = 0.7 ms
2.47.1	Case 47 - Impulsively Loaded Cylindrical Panel - Model and Material Properties
2.47.2	Case 47 - Deformed Shape of Crown Line
2.47.3	Case 47 - Vertical Deflection of Node 273 and 409
2.47.4	Case 47 - Deformed Shape at $X=6.28$ in.
2.47.5	Case 47 - Response at Time = 0.8 ms
2.47.6	Case 47 - Plastic Strain Distribution - View A
2.47.7	Case 47 - Plastic Strain Distribution - View B
2.48.1	Case 48 - Experimental Test Rig and Material Properties
2.48.2	Case 48 - Impulsively Loaded Circular Plate - Node Numbers
2.48.3	Case 48 - Impulsively Loaded Circular Plate - Element Numbers
2.48.4	Case 48 - Displacement in Y-Direction at Node 1 - Comparison of Theoretical and Experimental Results
2.48.5	Case 48 - Response at Time = 0.2 ms
2.49.1	Case 49 - Moment Load Test for Non-Linear Shell (Dynamic Relaxation) - Model and Load
2.49.2	Case 49 - Displacement of Node 13 in both X and Y Directions
2.49.3	Case 49 - Response at Time = 0.1 sec.
2.49.4	Case 49 - Response at Time = 0.4 sec.
2.50.1	Case 50 - Corner Supported Flat Plate Under Pressure Load (Explicit Solution) - H3DMAP Model
2.50.2	Case 50 - Vertical Deflection of Node 73
2.50.3	Case 50 - Response at Time = 0.02 sec.
2.51.1	Case 51 - Axial Impact of a 3-D Beam - Rigid Wall Model and Response of Node 104
2.52.1	Case 52 - Axial Impact of a 3-D Beam - Boundary Restrictions - Response of Node 104
2.53.1	Case 53 - Simply Supported Beam Model - Step Load at Mid-Span
2.53.2	Case 53 - Simply Supported Beam - Step Load at Mid-Span - Response of Node 72
2.54.1	Case 54 - Beam Model
2.54.2	Case 54 - Axial End Load on Cantilever Beam - Model and Response of Node 84
2.55.1	Case 55 - Dynamic Relaxation - Elastic-Plastic Material - Node 84 Displacement
2.55.2	Case 55 - Dynamic Relaxation - Elastic-Plastic Material - Element 20 Axial Stress
2.55.3	Case 55 - Dynamic Relaxation - Elastic Material - Node 84 Displacement
2.55.4	Case 55 - Dynamic Relaxation - Elastic Material - Element 20 Axial Stress
2.56.1	Case 56 - Geometric Non-Linear Beam Test Model
2.56.2	Case 56 - Geometric Non-Linear Beam Test - Response at Time: 3 ms.
2.56.3	Case 56 - Geometric Non-Linear Beam Test - Response of Node 132
2.57.1	Case 57 - Undamped Spring-Mass System with Initial Velocity - Model and Displacement Response
2.57.2	Case 57 - Undamped Spring-Mass System with Initial Velocity - Velocity Response
2.57.3	Case 57 - Theoretical Solution - Displacement vs. Time
2.57.4	Case 57 - Theoretical Solution - Velocity vs. Time
2.58.1	Case 58 - Undamped Spring-Mass System with Force Step Function
2.59.1	Case 59 - Undamped Spring-Mass System - Beat Response
2.60.1	Case 60 - Annular Element Testing (Explicit Solution)
2.61.1	Case 61 - Sandwich Model

List of Figures ... 4

2.61.2	Case 61 - Sandwich Model - Displacement Response
2.62.1	Case 62 - Tritiated Heavy Water Transportation Package
2.62.2	Case 62 - Overall Full Scale THWTP FSI Model Geometry
2.62.3	Case 62 - THWTP FSI Model Dimensions
2.63.3a	Case 62 - Exploded View of the Model
2.62.4	Case 62 - Deformed View on Plane (X=0.0) - Time = 0.0050 sec.
2.62.5	Case 62 - Analytical Results (Full Scale Model)
2.62.6	Case 62 - Test Results (1/4 Scale Model)
2.62.7	Case 62 - Hydrodynamic Stress Distribution - Time = 1 ms
2.62.8	Case 62 - Hydrodynamic Stress Distribution - Time = 2 ms
2.62.9	Case 62 - Hydrodynamic Stress Distribution - Time = 3 ms
2.62.10	Case 62 - Hydrodynamic Stress Distribution - Time = 5 ms
2.62.11	Case 62 - Hydrodynamic Stress Distribution - Time = 6 ms
2.62.12	Case 62 - Hydrodynamic Stress Distribution - Time = 7 ms
2.62.13	Case 62 - Effective Plastic Strain - Inner Side of Shell Elements
2.62.14	Case 62 - Effective Plastic Strain - Outer Side of Shell Elements
2.63.1	Case 63 - Fluid Hydrodynamic Material Testing - H3DMAP Model
2.63.2	Case 63 - Pressure Distribution in the Pipe at Time = 5.8 ms.
2.63.3	Case 63 - Pressure Distribution in the Pipe at Time = 10.77 ms.
2.63.4	Case 63 - Displacement of Free Surface - Node 84
2.64.1	Case 64 - Dynamic Relaxation Test - H3DMAP Model
2.64.2	Case 64 - Displacement Response of Free End (Node 84)
2.64.3	Case 64 - Stress Response of Free End (Element 20)
2.65.1	Case 65 - Pipe Whip Test - Model and Material Properties
2.65.2	Case 65 - Deformed Structures - Response at Time: 3.5 ms.
2.65.3	Case 65 - Effective Plastic Strain - Outer Shell - Time: 3.5 ms.
2.65.4	Case 65 - Effective Plastic Strain - Inner Shell - Time: 3.5 ms.
2.65.5	Case 65 - Von Mises Stress - Outer Shell - Time: 3.5 ms.
2.65.6	Case 65 - Von Mises Stress - Inner Shell - Time: 3.5 ms.
2.65.7	Case 65 - X-Displacement Time History of Node 1
2.66.1	Case 66 - Tritiated D ₂ O Transport Container Punch Test
2.66.2	Case 66 - Node Numbering
2.66.3	Case 66 - Node Numbering - Close-Up of Impact Area
2.66.4	Case 66 - Yield vs. Pressure Curve for Polyurethane Foam Model
2.66.5	Case 66 - Pressure/Volumetric Strain Curve for Polyurethane Foam Model
2.66.6	Case 66 - Deformed Geometry - Response at Time = 28 ms.
2.66.7	Case 67 - Vertical Displacement at the Periphery (Node 1474)
2.66.8	Case 66 - Impact Force Time History
2.66.9	Case 66 - Plastic Strain in Shell Element No. 10, Sampling Point No. 3 (see figure 2.66.3)
2.66.10	Case 66 - Hydrostatic Stress in Solid Element No. 417 (see figure 2.66.3)
2.66.11	Case 66 - Plastic Strain in Metal Sheet (Outer Shell) at Time: 28 ms
2.66.12	Case 66 - Plastic Strain in Metal Sheet (Inner Shell) at Time: 28 ms
2.66.13	Case 66 - Von Mises Stress in Foam at Time: 28 ms
2.66.14	Case 66 - Hydrostatic Stress in Foam at Time: 28 ms
2.67.1	Case 67 - Simplified Analytical Cone Model
2.67.2	Case 67 - H3DMAP Model
2.67.3	Case 67 - Impact Force vs. Time
2.67.4	Case 67 - Response at Time = 8 ms
2.67.5	Case 67 - Plastic Strain Distribution at Time = 8 ms

List of Figures ... 5

2.68.1	Case 68 - Problem Description for the Sphere Impact Analysis
2.68.2	Case 68 - Finite Element Model for the Sphere Impact Analysis
2.68.3	Case 68 - Displaced Shape of the Sphere 3 Milliseconds After Impact
2.68.4	Case 68 - Comparison of Final Deformation Between H3DMAP and Experimental Values
2.68.5	Case 68 - Time History Plot of Diametral Reduction for the Sphere Impact Analysis (Comparison Between H3DMAP and Experimental Values)
2.69.1	Case 69 - Scale Model RFTP Assembly
2.69.2	Case 69 - RFTP Computer Model
2.69.3	Case 69 - Exploded View of the RFTP Computer Model
2.69.4	Case 69 - Template for the RFTP Computer Model
2.69.5	Case 69 - RFTP 9m Drop Test Set-Up
2.69.6	Case 69 - RFTP After 9m Corner Drop Test - External
2.69.7	Case 69 - RFTP After 9m Corner Drop Test - Internal
2.69.8	Case 69 - Experimental RFTP Model Post-Impact Deformation
2.69.9	Case 69 - Deformed Geometry at 1 ms
2.69.10	Case 69 - Deformed Geometry at 2 ms
2.69.11	Case 69 - Deformed Geometry at 3 ms
2.69.12	Case 69 - Deformed Geometry at 4 ms
2.69.13	Case 69 - Deformed Geometry at 5 ms
2.69.14	Case 69 - Deformed Geometry at 6 ms
2.69.15	Case 69 - Deformed Geometry at 7 ms
2.69.16	Case 69 - Deformed Geometry at 8 ms
2.69.17	Case 69 - Deformed Geometry at 8 ms - View A
2.69.18	Case 69 - Deformed Geometry at 8 ms - View B
2.69.19	Case 69 - Deformed Geometry at 8 ms - View C
2.69.20	Case 69 - Deformed Geometry at 8 ms - View D
2.69.21	Case 69 - Hydrodynamic Stress Distribution at Time 1 ms
2.69.22	Case 69 - Hydrodynamic Stress Distribution at Time 2 ms
2.69.23	Case 69 - Hydrodynamic Stress Distribution at Time 3 ms
2.69.24	Case 69 - Hydrodynamic Stress Distribution at Time 4 ms
2.69.25	Case 69 - Hydrodynamic Stress Distribution at Time 5 ms
2.69.26	Case 69 - Hydrodynamic Stress Distribution at Time 6 ms
2.69.27	Case 69 - Effective Plastic Strain - Outer Side of Shell Elements - View A
2.69.28	Case 69 - Effective Plastic Strain - Outer Side of Shell Elements - View B
2.69.29	Case 69 - Effective Plastic Strain - Inner Side of Shell Elements - View A
2.69.30	Case 69 - Effective Plastic Strain - Inner Side of Shell Elements - View B
2.69.31	Case 69 - Time History of Impact Force in the Z-Direction
2.69.32	Case 69 - Comparison of Predicted and Experimental RFTP Deformation at the Plane of Symmetry
2.70.1	Case 70 - Diagram of Rolling Process
2.70.2	Case 70 - Assembled Views With Dimensions
2.70.3	Case 70 - Assembled Views of Finite Element Model
2.70.4	Case 70 - Contact Surface Set 1
2.70.5	Case 70 - Contact Surface Set 2
2.70.6 (a-d)	Case 70 - Progression of Roller Motion and Deformation
2.70.7	Case 70 - Assembled Finite Element Model
2.70.8	Case 70 - Effective Plastic Strain in Assembly
2.70.9	Case 70 - Effective Plastic Strain in Pressure Tube

List of Figures ... 6

2.71.1	Case 71 - Contact Algorithm Test - Momentum Transfer
2.71.2	Case 71 - Three Dimensional Indentor - Model Geometry
2.71.3	Case 71 - Three Dimensional Indentor - Deformation at Time = 0.00173 sec.
2.71.4	Case 71 - Three Dimensional Indentor - Deformation at Time = 0.00422 sec.
2.71.5	Case 71 - Shell-Solid Contact - Model Geometry
2.71.6	Case 71 - Shell-Solid Contact - Deformation at Time = 0.01 sec. (View A)
2.71.7	Case 71 - Shell-Solid Contact - Deformation at Time = 0.01 sec. (View B)
2.71.8	Case 71 - Discontinuous Surface Contact Model 1 - Undeformed Geometry
2.71.9	Case 71 - Discontinuous Surface Contact Model 1 - Deformation at Time = 0.007 sec. (View A)
2.71.10	Case 71 - Discontinuous Surface Contact Model 1 - Deformation at Time = 0.007 sec. (View B)
2.71.11	Case 71 - Discontinuous Surface Contact Model 2 - Undeformed Geometry
2.71.12	Case 71 - Discontinuous Surface Contact Model 2 - Deformation at Time = 0.001 sec.
2.71.13	Case 71 - Discontinuous Surface Contact Model 2 - Deformation at Time = 0.002 sec.
2.71.14	Case 71 - Discontinuous Surface Contact Model 2 - Deformation at Time = 0.006 sec.
2.72.1	Case 72 - Finite Deformation Simple Shear Test Using Numerical Algorithm
3.1	Euler-Bernoulli Straight Beam
3.2	Euler-Bernoulli Curved Beam
3.3	3-D Spring Element
3.4	Non-Linear Element
3.5	3-Dimensional Non-Linear Annular Gap Element (ITYPG=3)
3.6	Typical Force-Deflection Curves
3.7	Bilinear Degenerated Isoparametric Thin/Thick Shell Element
3.8	Linear Isoparametric Beam Element
3.9	Isoparametric 2-Dimensional/Axisymmetric Continuum Element
3.10	Isoparametric 3-Dimensional Hexahedron Continuum Element
3.11	Pressure Surface Definition
3.12	Isoparametric 3-Dimensional Pentahedron Continuum Element
3.13	Stress/Strain Measures - Elasto-Plastic Material Model
3.14	Pressure Versus Yield Stress Curve
3.15	Pressure/Volumetric Strain Equation of State with Compaction
3.16	Contact Algorithm Master/Slave Surface Definition
3.17	Roller Motion Definition



ontario hydro
research division

To Dr. D. Mills
Director of Research

H3DMAP VERSION 4
A GENERAL THREE DIMENSIONAL FINITE ELEMENT COMPUTER CODE
FOR LINEAR AND NON-LINEAR ANALYSES OF STRUCTURES:
USER MANUAL AND VERIFICATION CASES - VOLUME 1, REVISION 1

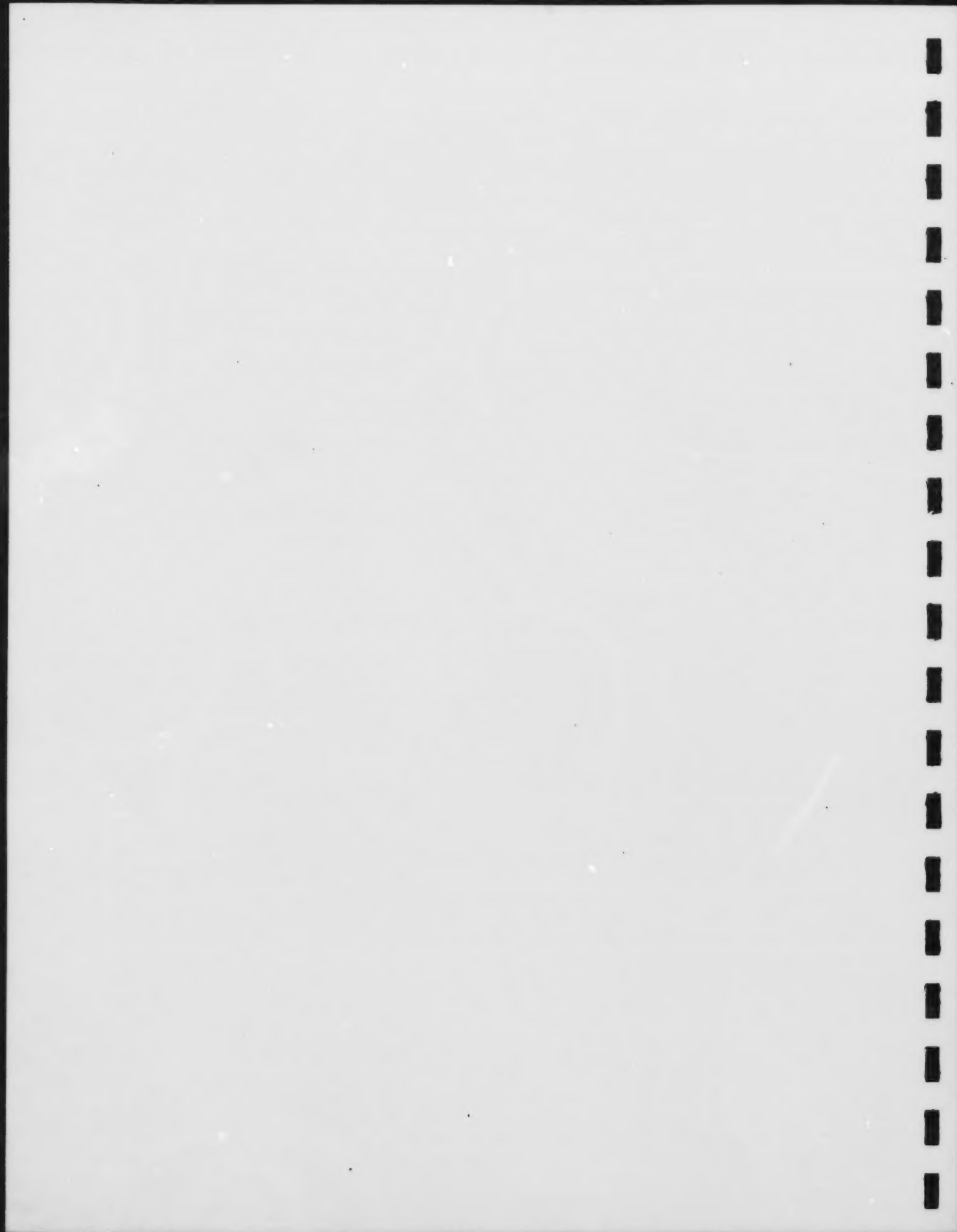
1.0 INTRODUCTION

The finite element computer code H3DMAP Version 4 is a general three dimensional structural analysis package that has been developed at Ontario Hydro to solve a wide variety of problems encountered by the corporation. H3DMAP is an acronym for **Hydro 3 Dimensional Matrix Analysis Program**. Previous versions of this code have been employed at Ontario Hydro to solve a number of problems related to reactor component design and licensing over the last number of years. The code has been extensively benchmarked and verified for a number of cases using both experimental results and classical solutions. The program is based on a highly modular architecture with simulation modules capable of handling the following procedures:

- static analysis of thermal and mechanical loads
- modal analysis
- frequency response
- response spectrum generation
- flow induced vibration
- seismic analysis (response spectrum method)
- buckling/stability analysis
- linear and non-linear geometric transient dynamic analysis using the implicit method
- geometric and material non-linear explicit transient dynamic analysis using the hydrodynamic formulation
- shock and wave propagation including interaction with hydrodynamic materials.

H3DMAP includes the following elements:

- straight and curved Euler-Bernoulli (cubic) beams
- isoparametric thin/thick beams or trusses
- pipes
- isoparametric thin/thick shells
- isoparametric 2-D/axisymmetric continuum
- isoparametric 3-D continuum
- spring elements
- analog elements



In addition, the program currently handles a variety of material models:

- elastic
- elastic-plastic
- hydrodynamic (e.g. fluid) with equation of state
- isotropic-hydrodynamic elastic-plastic with equation of state.

The last type of material can be used to model soil-like and foam-like materials. New material models are being developed and added on an ongoing basis. For finite deformation involving large material rotation, the principal axes algorithm is used/61,65/.

Among the salient features of H3DMAP are the flow induced vibration option and the non-linear explicit transient dynamic option for the finite deformation of solids.

The flow induced vibration option provides a means of evaluating the response of process equipment tubes to periodic wake shedding and turbulent flow excitation. Included in this option is the evaluation of critical flow velocities. This is used to determine the susceptibility of tubes to fluid/elastic instability. The implicit non-linear transient dynamic solution technique includes automatic time stepping with equilibrium iterations to ensure temporal convergence along with optimal performance. It has been used extensively in the prediction of both fretting wear forces arising from process equipment tube-to-baffle interaction under flow induced excitation and the structural interaction between pressure vessels and supports undergoing dynamic excitation.

The explicit analysis option is based on the hydrodynamic finite element formulation which is suitable for finite deformation problems. The material types currently available include elastic, elastic-plastic with isotropic hardening, general elastic-plastic hydrodynamic, and hydrodynamic models which are used in conjunction with equations of state to provide material models for fluids, soil and foams. A robust three-dimensional contact-impact algorithm/60/ is included to permit accurate modelling of bodies undergoing sliding at material interfaces. It is based on the master-slave slideline concept and provides significant code capabilities in the area of crashworthiness and manufacturing problems. Loading conditions include time dependent follower traction forces, concentrated nodal forces/moments, specified velocities and initial conditions such as velocities in impulsive problems. This approach makes it possible to model fluid-structure interaction problems. The explicit module is intended for problems involving impact, wave propagation and shock. In addition, a dynamic relaxation procedure has been included in H3DMAP which permits the solution of large non-linear static problems as a dynamic problem. The advantage here is that the explicit technique requires very little memory, compared to implicit methods, since the storage and solution of large sets of equations are avoided. Problems involving tens of thousands of degrees of freedom are feasible. Explicit techniques are well suited to impulsively loaded structural and crashworthiness problems involving large deformation and fluid-structure interaction. This option is currently used in the impact analysis of radioactive waste transportation containers and in the simulation of reactor component response.

The H3DMAP code architecture uses a form of dynamic storage allocation so that a single array controls the capacity. The scheme calculates address locations of the various arrays required for a specific problem so that only the necessary memory is utilized. The code is set to exploit the memory limits of the particular computer in which H3DMAP is installed. All coding is written in standard FORTRAN 77 and avoids the use of compiler extensions in order to remain device independent and portable. The modules and code structure are shown in Volume 1, Figures 1.1 to 1.13. The overall code capabilities are listed in Figure 1.14 of Volume 1.

H3DMAP is fully documented with comprehensive post-processing and graphics capabilities. Input data error checking is included for ease of use. The program is currently operational on DEC computers and workstations under both VAX/VMS and ULTRIX operating systems, Silicon Graphics and APOLLO computers under the UNIX operating system, and 386 based PC's under DOS. With minimum effort, the program can be ported to

other environments. Because of its modular structure, the code can also be easily modified to suit particular applications.

Volume 1 of this report provides the documentation of the verification cases used to validate the computer code and Volume 2 documents the user manual and typical input files required to run the verification cases.

2.0 VERIFICATION CASES

This section presents the verification cases used to benchmark H3DMAP. The names following the titles represent the computer file identification for the verification case.

2.1 Case 1 - Straight Beam Element

MAP1

A three-dimensional straight beam using the elements described in Figure 3.1 (Vol. 2) was modelled and analyzed. The configuration is shown in Figure 2.1.1. Nodal forces of 10 lbs (X-Force) and 50 lbs (Y- Force) were applied at node number 2. The boundary conditions at node numbers 1 and 3 were fully fixed in the local y-z plane and simply supported in the x-z plane. In addition to verifying the element, this model illustrates the use of the third point coordinate data (x_3, y_3, z_3) in the local element x-z plane. As shown in Figure 2.1.1, the computer model results for the deflections and bending moments at node number 2, agree with the calculated results obtained by using classical methods.

2.2 Case 2 - Curved Beam Element

MAP2

A three-dimensional curved beam using the element described in Figure 3.2 (Vol. 2) was modelled and analyzed. The configuration is shown in Figure 2.2.1. Nodal forces of 10 lbs (X-Force) and 50 lbs (Y- Force) were applied at node number 3. These forces result in in-plane and out-of-plane displacements and bending moments. The boundary condition at node number 1 was fully fixed in the local x-z plane. In addition to verifying the curved beam element, this model illustrates the use of the third point coordinate data (x_3, y_3, z_3) in the local element $x'-y'$ plane. The third point data is determined by the intersection of the tangents at the nodal points. As indicated in Figure 2.2.1, the computer program results agree with the results obtained by using classical methods.

2.3 Case 3 - Member Degree of Freedom Releases

MAP3

To illustrate the use of the member degree of freedom release option, a model incorporating three straight beam elements was used. The configuration of the model is shown in Figure 2.3.1. At node number 2 forces of 50 lbs (Y-Force) and 10 lbs (X-Force) were applied. The boundary conditions at node numbers 1 and 3 were fully fixed in the x-y plane and simply supported in the x-z plane. The boundary condition at node number 4 was fully fixed in the x, y and z directions. The member degree of freedom releases were the rotations about the local y and z axes at node number 2 (element number 3). Thus, element number 3 was uncoupled from elements 1 and 2 in the transfer of moments. The analytical results obtained by classical methods agree with the computer program results as shown in the table in Figure 2.3.1.

2.4 Case 4 - Piping Test Loop Eigensolution

LOOP1

For this case, the computer program results using the modal analysis option (LTYP=1) were compared to the experimental results for the primary heat transport pump test loop (Reference 17). In the experiment, several tests at different pressures, temperatures, and voids were performed to test the pump under several possible service conditions. During each of these tests, instrumentation recorded the hydraulic and structural response of the piping test loop. The analytical approach involved a dynamic analysis of the piping test loop under steady state excitation conditions.

The three dimensional piping test loop computer model is shown in Figure 2.4.1. Figures 2.4.2 and 2.4.3 show node numbers, element topology, and boundary conditions respectively. The computer model contains 54 nodal points and has 35 straight beam elements, 18 curved beam elements, and two spring elements. The piping is assumed to be linearly elastic and of uniform material (SA234 GR.WPB). In Figure 2.4.2, node numbers 1, 4 and numbers 2, 5 represent the pump support rods. Elements 3-6, 4-6, 5-6, 6-7 and 7-8 represent the pump components. Figure 2.4.3 shows the pump weight of 20,000 lbs and the motor weight of 23,000 lbs as concentrated masses. Similarly the weights of the piping, valves, and flanges were applied as shown in Figure 2.4.3. The weight of the water in the pipes was included in the model.

In the original analysis of the piping test loop, two boundary condition sets were utilized. The first set represented the idealized pipe restraint configuration of the as-built test loop. The second boundary condition set (Figure 2.4.3), was an approximation of the situation during the experimental modal analysis of the loop. As discussed in Reference 17, it is assumed that these boundary conditions are the most representative of the pipe support configuration. The information extracted from the experimental modal analysis, and the analytical frequency spectra at selected node numbers and global axis directions are given in the following table. It is apparent from this table that the analytical model is a good representation of the as-built piping loop from a structural dynamic standpoint. Figures 2.4.4, 2.4.5, 2.4.6 and 2.4.7 show the first, second, third and fifth mode shapes for the pump test loop.

Comparison of Analytical and Experimental
Model Analysis of the Piping Test Loop

Node No.	Direction	Experimental Frequency (Hz)	Computer Analysis Frequency (Hz)
3	Z	1.95	1.96
3	X	3.05	2.48
38	Y	11.63	13.05
38	Z	14.35	16.33

2.5 Case 5 - Seismic Analysis

MAP4

To illustrate seismic analysis by the response spectrum method option (LTYP=6), a three story building frame as outlined in Reference 6 and shown in Figure 2.5.1 was modelled and analyzed. The building frame is represented by a close-coupled system with three lumped masses and stiffnesses. Translations were permitted

in the x-axis direction at node numbers 2, 3 and 4. All the other translations and rotations in the global directions were eliminated.

As shown in Figure 2.5.1, the three natural frequencies obtained using the computer program agree with the frequencies presented in Reference 6. Also shown in the figure are the Root Sum of the Squares (RSS) accelerations and displacements at each node in the global x-axis direction.

2.6 Case 6 - Flow Induced Vibration

MAP5

To provide verification of the computer program for the flow induced vibration option (LTYP=5), an example from Reference 11 was analyzed. The model configuration, mode shapes, and results shown in Figure 2.6.1 are taken from Reference 11. The computer model analyzed is shown in Figure 2.6.2. The computer model consists of 30 nodal points and 29 straight beam elements. The periodic wake shedding resonant response was calculated by using a lift coefficient of 0.10. Comparison of the mode shapes of Figures 2.6.1 and 2.6.2 indicate excellent correlation. The following table compares relevant numerical results of the computer program H3DMAP and the Atomic Energy of Canada program "PIPEAU" (Reference 11).

**Comparison of Flow Induced Vibration
Analysis with AECL Chalk River Data**

Mode	Natural Frequency (Hz)	$\frac{V_r}{V_{r_{crit}}}$	Resonant Amplitude (μm)	Strouhal Number
1	17.5 (17.65)*	.701 (.69)	89 **	.12 **
4	26.0 (26.46)	.668 (.66)	242 **	1.73 **
8	41.0 (42.66)	1.369 (1.33)	1790 (1850)	.274 (.29)

* Pipeau results shown in brackets.

** Results not available.

2.7 Case 7 - Linear Transient Response

MAP8

In Case 7, a linear transient dynamic analysis was performed using the implicit transient dynamic option. The model configuration is shown in Figure 2.7.1. Node numbers 2 and 3 had masses of 2 lbm and 1 lbm respectively, and the value of the spring constant (at node number 3) was 2 lb/in. Free translation boundary conditions existed at node numbers 2 and 3 in the x and y directions, and all the other degrees of freedom were fully fixed. A 10 lbf step function was applied at node number 3. Damping was neglected in the system. This

model is discussed further in Reference 3. Figure 2.7.1 shows a plot of the system responses for nodes 2 and 3 in the global Y-direction. Good agreement between numerical and classical solutions is observed.

2.8 Case 8.1, 8.2 - Initial Velocity Response

To demonstrate the use of the variable time stepping scheme described in References 14 and 47, an initial velocity response problem was studied. The problem was taken from Reference 20 and consists of a straight beam subjected to an initial velocity profile as shown in Figures 2.8.1 and 2.8.2. In Figure 2.8.1, the spring stiffness, K , equals $.60 \times 10^8$ lb/in, and in Figure 2.8.2 the spring stiffness was $.60 \times 10^6$ lb/in. The analytical results were calculated as outlined in Reference 20. The computer results obtained by using H3DMAP are seen to be in excellent agreement with the analytical results. It is of interest to note that the converged solution was obtained in relatively few time steps (i.e. $NT=21$ for $K=.60 \times 10^8$ lb/in and $NT=19$ for $K=.60 \times 10^6$ lb/in).

2.9 Case 9 - Non-Linear Transient Dynamics (Boundary Element)

MAP9

A comparison with a documented case was made to demonstrate non-linear boundary elements and the efficiency and accuracy of the non-linear transient dynamic analysis option described in References 14, 43 and 47. The geometry for the boundary element is given in Figure 3.4. The model shown in Figure 2.9.1 represents a control rod drive housing which is subjected to sinusoidal support acceleration at 5 Hz. Mass proportional loading was used to simulate the support acceleration. The housing deflection is limited by a linear annular stop. Reference 18 refers to a solution using the direct integration method and a constant time step at $Dt=.0001$ sec (700 time steps) without equilibrium iterations. A run was made with the scheme presented, using $TOLD=2,000$ lbs and $PNIT=.05$, and a converged solution as shown in Figure 2.9.1 was obtained in 89 time steps. This solution is in agreement with the results of Reference 18. Thus, an accurate solution was obtained with a much smaller number of time steps. Observations indicate that the time step is cut back once an impact is detected and grows steadily when convergence of the iteration is assured. Further discussion is given in References 14 and 43.

2.10 Case 10 - Non-Linear Transient Dynamics (Inter-Element)

MAP10, XMAP10

The use and verification of the non-linear elements in an impact sequence are illustrated in this case which is shown in Figure 2.10.1. The geometry for the interconnecting element is given in Figure 3.4. The element can be used to represent general non-linear behaviour which is described in terms of the piecewise linear force deflection curve shown in Figure 2.10.1. The analytical results were obtained classically by considering the conservation of momentum:

$$(1) \quad M_1 V_1 + M_2 V_2 = M_1 v_1 + M_2 v_2 \quad V_i = \text{velocity in the initial state}$$

$$M_1(V_1 - v_2) = M_2(v_2 - V_2) \quad v_i = \text{velocity in the final state}$$

$$V_1 \neq v_1 \quad V_2 \neq v_2$$

$$(2) \quad \frac{1}{2}(M_1 V_1^2 + M_2 V_2^2) = \frac{1}{2}(M_1 v_1^2 + M_2 v_2^2)$$

$$V_1 + v_1 = V_2 + v_2$$

The H3DMAP results using the implicit operator (LTYP=2) are plotted in Figure 2.10.1 and agree with the analytical calculations. The plot illustrates the transfer of momentum between node numbers 1 and 2. The same results were obtained using the explicit operator (LTYP=8).

2.11 Case 11 - Non-Linear Transient Dynamics (Annular Element)

MAP13

The purpose of this case is to demonstrate the application of the non-linear annular element described in References 14 and 47. The model shown in Figure 2.11.1 uses three straight beam elements and a non-linear annular element at node number 4. The geometry for the annular element is given in Figure 3.5. Harmonic loading is applied at node number 3 as shown in Figure 2.11.1. The orbit of node number 4 in the annulus is shown in Figure 2.11.2.

2.12 Case 12 - Fretting Wear Force Prediction (Benchmark with AECL)

AECLBENCH2

Interest in the numerical scheme presented in References 14, 43 and 47 resulted in a request by Atomic Energy of Canada Chalk River Laboratories to perform a benchmark analysis. The object of the benchmark was to provide analytical results for a moderately complex model which can be used to compare the Ontario Hydro computer code and work underway at AECL. The ultimate aim is to provide a combined experimental/analytical approach for predicting wear rates from which a time to through-wall failure of process equipment tubing due to fretting wear can be obtained. The model used in the benchmark is shown in Figure 2.12.1. The diametral clearance was .032 inches and the value for the linear support stiffness was 10^5 lb/in since this value is representative of the values observed in practice. It should be noted that the H3DMAP computer code has the capability of representing quite arbitrary force-deflection relations for non-linear elements and as a result, simulation models can incorporate non-linear contact theories, squeeze film effects, etc.

In the computer run, no tuning of the model was made, and the automatic features of the scheme were invoked. Modal damping ratios of the order .005 were used in the form of Rayleigh proportional damping. As seen in Figure 2.12.2 the displacements of the tube at node number 25 are arrested at approximately .016 inches which is the magnitude of the radial gap. Chattering of the tube against the support is observed and is representative of what occurs in practice. The impact force time history at node number 25, given in Figure 2.12.3, is presented in the time interval from .15 sec to .25 sec where a quasi-steady state condition has been achieved. As seen, peak interaction forces of approximately 45 lbs occur between the tube and support. In References 14 and 27, a detailed discussion of the use of these forces in the fretting wear of process equipment tubes is presented. In Figure 2.12.4, the corresponding velocity time history indicates peak response at times of impact. The plot of impact force versus time between .17 sec and .25 sec obtained from AECL work is shown in Figure 2.12.5. Comparison with Figure 2.12.3 demonstrates good correlation.

2.13 Case 13 - Sine Beat Verification

SINEBEAT

This case illustrates the beat phenomena that occurs when a system is subjected to an external forcing frequency that is slightly greater than the structure's natural frequency. The computer code model, material properties, and displacement time history is shown in Figure 2.13.1. As observed in this figure, the computer code results compare very well with classical results.

2.14.1 Case 14.1 - Simple Frame Stability Analysis

BUCKLE4

A simple frame loaded vertically, as outlined in Reference 26, was modelled and analyzed to determine the lowest critical buckling mode. The 7 noded/6 element model is shown in Figure 2.14.1. Initially a static analysis was performed to obtain the linear element loads. In this analysis node number 1 was fully fixed, lateral motion constrained, and a global Y-direction (axial) force of 500 lb was applied. For the buckling analysis, lateral constraints were released, but node number 7 was fixed in the global Y-direction. Figure 2.14.1 shows excellent agreement between the classical fundamental eigenvalue and the H3DMAP computer code results.

2.14.2 Case 14.2 - Euler Column Buckling Analysis

BUCKLE3, ISOBTEST3

A long, slender column loaded in pure compression along the centroidal axis was modelled and analyzed to determine the condition of unstable equilibrium. The 9 noded/8 element model is shown in Figure 2.14.2. Node number 1 is fully fixed with all other nodes unconstrained, and a 50 lb global Y-direction force is applied at node number 9. A static analysis is first performed to obtain the initial column forces. A buckling analysis is then executed to obtain the fundamental eigenvalue. Figure 2.14.2 shows good correlation for the first mode between the classical solution and the H3DMAP computer code results using both the cubic beam and linear isoparametric beam.

2.15 Case 15 - Lateral Stability of a Cantilever Beam

BUCKLE1

A cantilever beam of constant cross-section loaded with a vertical shear at the free end was modelled and analyzed to determine the critical buckling eigenvalues. The critical buckling condition arises when the deflected shape in the vertical plane is unstable and lateral buckling occurs. Twenty equal beam elements were used in the modelling for the H3DMAP solution (Figure 2.15.1). Node number 1 is fully fixed with all other nodes unconstrained and a 50 lb global Y-direction force is applied at node number 21. A static analysis was performed initially, followed by the buckling analysis. Figure 2.15.1 shows that the H3DMAP computer code results for the 1st, 2nd and 3rd modes are in good agreement with the analytical solutions. In addition, the results were compared with the MSC/NASTRAN computer code and as observed excellent agreement has been obtained.

2.16.1 Case 16.1 - Straight Beam Element: Thermal Growth Test

MAP15

A three-dimensional straight beam, subjected to a uniform axial temperature gradient, and using the elements described in Figure 3.1 was modelled and analyzed. The model configuration is shown in Figure 2.16.1 where node number 1 is fully fixed and a temperature difference of 100°F exists across all the elements. This analysis shows that the H3DMAP computer model results for the maximum displacement compares well with the calculated results obtained by using classical methods.

2.16.2 Case 16.2 - Straight Beam Element: Thermal Load Test

MAP14

For this case, Figure 2.16.2, both ends of the model are fully fixed and a temperature difference of 100°F exists across all the elements. The computer model results show good agreement with the analytically calculated compressive forces.

2.17 Case 17 - Frame Structure - Mixed Elements

MAP16

The H3DMAP computer code was used to analyze a two-dimensional frame structure which incorporates beam and truss elements, rotational and translational springs, and temperature effects. The configuration is shown in Figure 2.17.1. A temperature gradient is applied at element number 3 and a static 1000 lb (global Y) load is applied at node number 3. The boundary conditions at node numbers 1 and 4 are fully fixed in all directions and node numbers 2 and 3 are free to move in the x, y and R_z (rotation) degrees of freedom. As indicated in Figure 2.17.1, the computer code results agree with the analytical results obtained using classical methods.

2.18 Case 18 - Curved Beam Element - Thermal Load Test

MAP17, MAP18

A three-dimensional curved beam, subjected to a uniform temperature gradient, and using the elements described in Figure 3.2 was modelled and analyzed. This configuration is shown in Figure 2.18.1. The boundary condition at node number 1 is fully fixed. Figure 2.18.1 illustrates that the global displacements computed by H3DMAP agree with the results obtained using classical methods.

2.19 Case 19 - Fretting Wear Force Prediction

MAP19

One case which represents a situation of tube to tube-support plate interaction is the work described in Reference 29. The experimental model used in Reference 29 consisted of a tube 0.99 m long, fully clamped at each end. A baffle plate and two impact transducers were located at midspan with a radial clearance of 0.127 m between tube and baffle as shown in Figure 2.19.1. The periodic excitation forces were applied at four discrete points along the span as shown. The data from Reference 29 was used to set up the equivalent analytical model shown in Figure 2.19.1.

The comparison of the experimental and analytical impact force and displacement time histories at the baffle are presented in Figure 2.19.2. As observed, excellent agreement between experiment and prediction is achieved. Chattering is seen in the displacement time history due to the strong impact. This phenomenon has been observed in many problems of this type. When the tube impacts the support plate, it rebounds until it is arrested by the excitation force. If the magnitude of the forcing function at impact is sufficiently large to overcome this motion, chattering results.

2.20 Case 20 - Pipe Whip Analysis

MAP26

The use of the method to obtain solutions to pipe whip problems was investigated by comparing the results of the well documented problem of Reference 30 with those obtained using H3DMAP. The model shown in Figure

2.20.1, consists of a cantilevered pipe subjected to an instantaneous blowdown force of constant amplitude. A clearance exists between the restraint and the pipe at the location of load application. The restraint is modelled using elastic perfectly plastic material properties. As indicated in Figure 2.20.1 agreement with the results of Reference 30 for a gap = 1.5" is excellent. A converged solution was obtained with as little as 33 time steps using the auto-time step algorithm as compared with 300 constant time steps used in Reference 30.

2.21 Case 21 - Single Degree of Freedom Study

MAP25, XMAP25

Initial study of the solution scheme discussed in References 14 and 47 was utilized in a single degree of freedom model as shown in Figure 2.21.1. This problem was solved analytically by considering a series of piecewise linear systems. The initial conditions were used to obtain a solution at the time the gap closed. The initial velocity $V_0 = 10$ in/sec was selected based on the system's natural frequency to ensure that the gap would close. The new system which includes the gap stiffness is solved using the response previously calculated at the time the gap closed as initial conditions. The problem was solved numerically with H3DMAP using both the implicit and the explicit operator. The table below indicates good agreement between analytical and computer results.

	Closure Time (sec)	Maximum Response Time (sec)	Maximum Response (in)	Maximum Response Force (lbs)
Classical	0.1009	0.1499	1.3058	0.3058×10^5
Numerical Computer Implicit Solution	0.1009	0.150	1.3053	0.3053×10^5
Explicit Solution*	0.1008	0.154	1.3037	0.3037×10^5

* Time step is coarse which explains slightly different results.

For the implicit solution, the tolerances specified for the problem were PNIT = .02 for iteration convergence and TOLD = 3000 lbs for the maximum allowable residual. The residual tolerance chosen is approximately 10% of the maximum impact force. Results were accurate for higher values of both PNIT and TOLD up to PNIT = .1 and TOLD = 5000 lbs. In Figure 2.21.1, both response and variation of time step with time are shown for the implicit solution (the results are quite similar for the explicit solution). At impact, the time step reduces, then gradually increases. At time = .18 sec, the time step reduces slightly, then continues increasing. This occurs because the convergence tolerance PNIT governed in this case causing the increase of time step based on the residual size to be overridden.

The parameters of the problem were varied to study the effect of numerical conditioning. In the most extreme case (i.e. $K_{GAP} = 10^{15}$ lb/in and $K=0$), the results indicated high accuracy.

2.22 Case 22 - Dynamic Relaxation Effects

GRAVTEST

This analysis was performed to verify the gravitational load and nodal force effects in H3DMAP. Three dimensional straight beams were used and the model configuration is shown in Figure 2.22.1. Also note in this figure the steady state response due to the gravitational loading and the static response of the structure due to

a nodal force applied at node 2. Initially, the beam is allowed to sag under its own weight until a steady state condition is reached at $t=0.06$ seconds (Figure 2.22.1). At $t=0.08$ seconds, a positive 25 lb force (Global Y direction) is applied at node number 2 which lifts the structure. A large mass damping coefficient of 1000 is introduced in the analysis to provide the dynamic relaxation. Non-linear boundary elements are present at node numbers 1 and 3 and the value for the linear support stiffness is $K=10^6$ lb/in. All out-of-plane translations and rotations were fixed.

The deflection of the three-dimensional beam at midspan due to the gravitational effects is given by $y = PL^3/48EI=0.00167$ inches. This compares well with the H3DMAP result shown in Figure 2.22.2. The equation of motion for this linear system will be in the form:

$$M\ddot{x} + C\dot{x} = P \quad C = \beta M = 1000M$$

$$M\ddot{x} + 1000 M\dot{x} = P$$

$$\ddot{x} + 1000 \dot{x} = \frac{P}{M} \quad ; \quad \frac{P}{M} = \frac{25-30}{20/386.4} = 96.6$$

The solution to the above equation has two parts. To solve for the homogeneous solution, set $P/M = 0$, then

$$\ddot{x} + 1000 \dot{x} = 0 \quad (1)$$

Assume a solution of the form

$$x = e^{mt} \quad (2)$$

$$\dot{x} = me^{mt} \quad (3)$$

$$\ddot{x} = m^2e^{mt} \quad (4)$$

Substituting (2), (3) and (4) into (1)

$$(m^2 + 1000m) e^{mt} = 0 \quad ; \quad e^{mt} \text{ cannot equal zero,}$$

$$\therefore m = -1000$$

Thus,

$$x = C_1 e^{-1000t} + C_2$$

The particular solution is obtained by assuming a solution where:

$$y = A_0 t$$

where $1000 A_0 = \frac{P}{M}$

$$A_0 = \frac{P}{1000 M}$$

Therefore:

$$y = \frac{P}{1000 M} \cdot t$$

Thus, the equation of motion is given by:

$$x = C_1 e^{-1000t} + C_2 + \frac{P}{1000 M} \cdot t$$

Differentiating gives:

$$\dot{x} = -1000 C_1 e^{-1000t} + \frac{P}{1000 M}$$

$$\ddot{x} = 10^6 C_1 e^{-1000t} = 0$$

For constant velocity, it is found that,

$$C_1 = 0$$

Therefore:

$$\dot{x} = \frac{P}{1000 M} = \frac{5}{1000 \cdot \frac{20}{386.4}} = 0.0966 \text{ in/s}$$

This result agrees with the final steady state velocity predicted by H3DMAP as seen in Figure 2.22.2.

2.23 Case 23 - Thin Cylindrical Shell Roof Analysis

SHEL1

A three dimensional cylindrical shell roof analysis was carried out using the bilinear degenerated isoparametric shell element described in References 47 and 57. The configuration of the 1/4 model (4 x 4 mesh) is shown in Figure 2.23.1. As shown in the Figure, the roof is supported at both ends on rigid diaphragms. Symmetric boundary conditions were applied along the X and Y symmetry planes. The longitudinal displacement (global) at the diaphragm (y=0) and the centreline vertical displacement (y=L/2) from the exact shallow shell theory (Reference 33) and from H3DMAP are shown in Figure 2.23.2. Excellent agreement is observed. The corresponding bending and membrane stresses in the longitudinal and transverse direction are shown in Figure 2.23.3.

2.24 Case 24 - Cylinder Under Internal Pressure

SHEL4, SHEL4A

A thin cylinder subjected to 1000 psi internal pressure was modelled and analyzed to determine the maximum displacement and stress values. A 1/4 model of the cylinder using a 1 x 4 mesh and a 1 x 8 mesh was developed for the analysis. The shell thickness used in both analysis was 0.1 inches and axisymmetric boundary conditions were applied. The H3DMAP predictions of displacement and stress show good agreement with the classical solutions ($\sigma = Pr/t$, $\delta = Pr^2(2-\mu)/2tE$) as shown in the table in Figure 2.24.1.

2.25 Case 25 - Simply-Supported Square Plate Analysis

2.25.1 Case 25.1 - Concentrated Load at Plate Centre

SHEL2, SHEL2A

To determine the bilinear shell element's response to static loading, a simply supported square plate was modelled and analyzed. The 1/4 model configuration is shown in Figure 2.25.1. Two meshes were developed, a 2 x 2 mesh and a 4 x 4 mesh. Symmetric boundary conditions were enforced along the symmetry axes and a concentrated 125 lb (global Z direction) force was applied at node number 21. The H3DMAP computer code results are in good agreement with those calculated by using classical theory (Reference 40) as shown in Figure 2.25.1.

2.25.2 Case 25.2 - Uniform Pressure Load

SHEL3, SHEL3A

The same simply supported square plate as shown in Figure 2.25.1 was used in this analysis except it was subjected to a uniform pressure load of 10 lb force (Global Z direction). A 2 x 2 mesh and a 4 x 4 mesh were modelled and analyzed and their results are shown in Figure 2.25.1. The H3DMAP maximum displacement and stress results agree with the calculated results obtained by using classical methods found in Reference 40.

2.25.3 Case 25.3 - Dynamic Modal Analysis

SHEL8

For this case a full model was used in the computer code modal analysis of a flat plate. As shown in the table in Figure 2.25.1., the five natural frequencies obtained by using the program agree with the frequencies calculated by classical methods found in Reference 41. Figures 2.25.2, 2.25.3, 2.25.4 and 2.25.5 show the first, second, fourth, and fifth mode shapes respectively for the flat square plate. Mode shape three is not included since it is the same as the second mode shape.

2.25.4 Case 25.4 - Clamped Square Plate Under Thermal Gradient

SHEL5

A 1/4 model (4 x 4 mesh) was analyzed to determine the effects of thermal loading on a clamped square plate. Figure 2.25.1 shows the model configuration used with the exception that the perimeter nodes (1 to 5 and 10, 15, 20, 25) are now fully fixed. Symmetric boundary conditions were also enforced along the symmetry axes. The coefficient of thermal expansion used in this analysis is 5.0×10^{-6} in/in/°F and a temperature gradient of 100

degrees is applied to all nodes. As shown in Figure 2.25.1, the program results compare well with the analytical solution $\left(\sigma_{max} = \frac{\alpha \cdot E \cdot \Delta T}{2(1-\mu)} \right)$.

2.25.5 Case 25.5 - In-Plane Expansion

SHEL6

For the in-plane expansion analysis, the 1/4 model (4 x 4 mesh) shown in Figure 2.25.1 was used. An ambient temperature of 100 degrees was specified and a coefficient of thermal expansion of 5.0×10^{-6} in/in/°F was used. Node number 1 was simply supported and node numbers 2 to 5 were fixed in the global X, Z and R_x directions. The remaining node numbers were fixed in the out-of-plane directions (i.e. Z displacement and R_x rotation). The table in Figure 2.25.1 shows that the computer program results agree with the analytical results obtained by classical methods ($\delta = \alpha L \Delta T$).

2.26 Case 26 - Clamped Hyperbolic Paraboloid Shell

SHEL10, SHEL10B

In this verification case, a 1/4 model was used to determine a hyperbolic paraboloid shell's static response to a uniform normal pressure of 0.01 lb/in². The model configuration is shown in Figure 2.26.1. For the full model, node numbers 9, 18, 27, 36, 45, 54, 63 and 72 to 81 are fully fixed. Nodes along the Y symmetry plane are fixed in the global Y and R_y directions and nodes along the X symmetry plane are fixed in the global X and R_x directions. The static deflection along the centre is shown in Figure 2.26.2 and is compared to the solution reported in Reference 33. H3DMAP results demonstrate excellent agreement with the finite difference solution.

2.27 Case 27 - Linear Dynamic Response of a Spherical Shell

SHEL7

A comparison with documented cases, References 36 and 38, was made in the case of a clamped shallow spherical cap under a concentrated force applied as a step load. A quarter shell model was developed due to the symmetry of both geometry and loading and the model configuration is shown in Figure 2.27.1. A 25 lb force step function was applied to the top of the spherical cap and damping was neglected in the system. The shell thickness used in this analysis was 0.01676 inches. The time span for the model analysis was 0.0006 seconds and an initial time step of 2.2×10^{-6} seconds was chosen. H3DMAP displacement time history results at node number 1 are compared with those reported in References 36 and 38 and are found to be in agreement as shown in Figure 2.27.2.

2.28 Case 28 - Pinched Circular Cylinder Test Case

SHEL9, SHEL9A, SHEL9B, SHEL9C

A circular cylinder with a midspan concentrated load was modelled in order to test the shell formulation. The 1/4 model (90 degree section) configuration is shown in Figure 2.28.1. Two meshes (4 x 4 and 4 x 6) were developed for this analysis. For the thin cylinder case, the shell thickness was $t = 0.01548$ inches and a midspan load of 0.1 lb was applied. The thicker cylinder had $t = 0.094$ inches and a midspan load of 100 lb. The ends of the circular cylinder are free and symmetric boundary conditions are applied along the global Y and Z symmetry planes. The H3DMAP results shown in Figure 2.28.1 are in agreement with the results reported in Reference 31 and with those calculated values for a thin cylinder in Reference 40.

2.29 **Case 29 - Stability of a Simply Supported
Square Plate Uniformly Compressed in One Direction**

SHEL11, SHEL11A

This problem consists of a thin, square elastic plate, which is compressed in one direction as shown in Figure 2.29.1. Two 1/4 model configurations, a 4 x 4 mesh and an 8 x 8 mesh were used in the analysis. All the perimeter nodes were fixed in the out-of-plane translation direction and simply supported boundary conditions (X- translation direction fixed) were enforced on node numbers 1 through 9. Applied to node numbers 73 and 81 were 37.5 lb forces (negative global X-direction) and 75 lb forces were applied to node numbers 74 through 80.

The critical buckling load is obtained by using an eigensolution. In Reference 26 the critical value of compressive stress σ_{cr} is:

$$\sigma_{cr} = \frac{4\pi^2 E}{12(1-\mu^2)} \left[\frac{h^3}{b} \right]$$

and $\lambda = \text{eigenvalue} = \frac{\sigma_{cr}}{\sigma}$. H3DMAP eigenvalue results for the 4 x 4 and 8 x 8 mesh are 491 and 473 respectively which is in good agreement with the theoretical eigenvalue of 471. Figure 2.29.2 is a plot of the fundamental buckling mode.

2.30 **Case 30 - Euler Shell Buckling Analysis**

SHEL14

This verification case is similar to case 2.14.2 except shell elements instead of straight beam elements are used in the model configuration, as shown in Figure 2.30.1. The shell element model was loaded in pure compression at one end. Twenty shell elements were used in the analysis, with a shell thickness of 0.05 in. At node numbers 21 and 42, a 25 lb force (negative global Y- direction) was applied. Node numbers 1 and 22 are fully fixed and all out of plane rotations (R_z) are fixed. The resulting fundamental eigenvalue of 0.129 is the same result produced by H3DMAP in case 12.14.2. The corresponding buckling mode is shown in Figure 2.30.2.

2.31 **Case 31 - Calandria Tube Shell Buckling Analysis**

SHEL12A

In this analysis a 20 element, 1/4 model was developed as shown in Figure 2.31.1. The shell thickness used was 0.061 in. and the calandria tube was subjected to an external pressure of 1.0 psi. Node numbers 1 and 22 are free in the global Y translational direction only and node numbers 21 and 42 are free in the global X translational direction. The H3DMAP predicted a fundamental eigenvalue of 55.93 as compared to the theoretical value of 49.69 determined by using

$$P_{cr} = \frac{Eh^3}{4(1-\mu^2)R^3} \quad (\text{Ref. 26})$$

This verification case was also done using the NASTRAN computer code and a fundamental eigenvalue of 55.72 was obtained. The corresponding buckling mode is shown in Figure 2.31.2.

2.32 Case 32 - Corner Supported Plate

CORNSHEL

This problem is defined in Figure 2.32.1. This case demonstrates the use of the stabilization matrix derived for the element. Without it, the zero energy mode (w-hourglass mode) leads to erroneous results. A relatively coarse (4 x 4) mesh is used in order to demonstrate the efficiency of the element. As seen in Figure 2.32.1, a quarter model representing the problem was used by applying symmetrical boundary conditions. Only the corner node (i.e. node 5 in the H3DMAP model) is supported in the translation degree of freedom. The plate is subjected to a pressure load. A comparison of displacements along the centreline with those obtained using classical theory and the commercial computer code ABAQUS were made. As seen in Figure 2.32.1 good agreement is achieved. It is of interest to note that a solution could not be obtained using the 4-noded shell element in ABAQUS version 4.5 since it exhibited the grossly oscillatory behaviour which is characteristic of an unstabilized element. Thus, the ABAQUS higher order 8-noded quadratic shell element was used in this comparison. The results indicate that the H3DMAP shell element is quite efficient since the H3DMAP model equivalent to the ABAQUS model for this problem would consist of an 8 x 8 instead of the 4 x 4 mesh which was used. A detailed discussion of hourglass modes in shell elements is given in Reference 57.

2.33 Case 33 - Isoparametric Beam Element Test

2.33.1 Case 33.1 - Concentrated Load at Centre

ISOBTEST1

To determine the isoparametric beam response to static loading, a model similar to that used in case 1 was analyzed. The model consists of 9 nodes and 8 elements as shown in Figure 2.33.1. The boundary conditions at nodes 1 and 9 were fully clamped in the X- Y plane and simply supported in the X-Z plane. As seen in Figure 2.33.1 the H3DMAP results indicate good agreement with classical theory.

2.33.2 Case 33.2 - Thermal Expansion

ISOBTEST1A

A model similar to that used in the previous section for mechanical loads was again utilized for the free thermal expansion test except that node 9 was released in the x-direction. A uniform temperature difference of 100°F was applied to all nodes. As seen in Figure 2.33.1, the results are in good agreement with those obtained using classical methods.

2.33.3 Case 33.3 - Dynamic Model Analysis

ISOBTEST1B

For this case, a model identical to that used for the mechanical loading in Case 33.1 is considered. The first three natural modes of vibration were determined using H3DMAP. As seen in Figure 2.33.1, the natural frequencies obtained are in good agreement with the natural frequencies calculated by classical methods in Reference 41.

2.33.4 Case 33.4 - Isoparametric Beam Column Buckling

ISOBTST3

A relatively long slender beam loaded in compression along the centroidal axis was modelled as shown in Figure 2.14.2. This case is similar to Case 14.2, with the exception that linear isoparametric beam elements are used instead of the Euler-Bernoulli beam elements. Initially, a static analysis is performed to obtain the initial column stresses. A subsequent eigenvalue stability analysis yields the fundamental eigenvalue which is a multiplier of the applied load vector to yield the buckling load. Figure 2.14.2 indicates excellent agreement of the H3DMAP Euler-Bernoulli and linear isoparametric beam results with the classical solution.

2.34 Case 34 - Cantilever Beam

AXIAL 2D and AXIAL 3D series

The simple model shown in Figure 2.34.1(a) was used to test the 2D continuum element under various conditions. First, it was subjected to an axial end load of 40,000 lbs. Then, thermal growth was verified using a 100°F temperature gradient. The same temperature gradient problem was run again fixing the free end to verify thermal stresses.

Finally, a similar model was used to verify Euler buckling. The beam was 0.5 inches wide and 10 inches long, with 10 elements across the width and 50 elements along the length (Figure 2.34.1(b)).

For the 3-D continuum element, the same models and test cases as the 2-D continuum element testing were redone in 3-D. The model used for axial load, thermal growth and thermal stress are shown on Figure 2.34.2(a), and the model used for buckling is shown on Figure 2.34.2(b).

For both the 2-D and 3-D elements, H3DMAP results are in good agreement with the classical solutions as shown in the following table.

Test Case	Case Characteristics	Classical Solution	H3DMAP 2-D Element	H3DMAP 3-D Element
Axial end load	F=40,000 lbs.	$\sigma_x = 20,000 \text{ psi}$ $\epsilon_x = 6.897 \times 10^{-4} \text{ in/in}$ $\Delta_{x_{\text{free end}}} = 6.897 \times 10^{-3} \text{ in}$	$\sigma_x = 20,000 \text{ psi}$ $\Delta_x = 6.89 \times 10^{-3} \text{ in}$	$\sigma_x = 20,000 \text{ psi}$ $\Delta_x = 6.89 \times 10^{-3} \text{ in}$
Thermal growth	$\Delta t = 100^\circ\text{F}$ One fixed end + one free end	$\epsilon_x = 6.5 \times 10^{-4} \text{ in/in}$ $\Delta_{x_{\text{free end}}} = 6.5 \times 10^{-3} \text{ in}$	$\Delta_x = 6.5 \times 10^{-3} \text{ in}$	$\Delta_x = 6.5 \times 10^{-3} \text{ in}$
Thermal stress	$\Delta t = 100^\circ\text{F}$ Both ends fixed	$\sigma_x = 18,850 \text{ psi}$	$\sigma_x = 18,850 \text{ psi}$	$\sigma_x = 18,800 \text{ psi}$
Euler buckling	Static load: 5,000 lbs.	$\gamma = 1.4908$	$\gamma = 1.58$	$\gamma = 1.62$

2.35 Case 35 - Thin Wall Vessel

THIN_AXI_PRESSURE
THIN_AXI_THERMAL_GROWTH

The thin wall vessel shown on Figure 2.35.1 was tested with an internal pressure, and also for thermal growth. The following table compares H3DMAP results with the classical solution.

Test	Classical Solution	H3DMAP
Internal pressure P = 1000 psi	$\sigma_{\theta} = 20,000$ psi	$\sigma_{\theta} = 20,000$ psi
Thermal growth (R = 2.05") $\Delta T = 100^{\circ}\text{F}$	$\Delta u_R = 0.00133$ in	$\Delta u_R = 0.00133$ in
(Y = 10")	$\Delta u_Y = 6.5 \times 10^{-3}$ in	$\Delta u_Y = 6.5 \times 10^{-3}$ in

2.36 Case 36 - Thick Wall Vessel

THICK_PRESSURE
THICK_THERMAL_GROWTH
THICK3D_PRESSURE
THICK3D_THERMAL_GROWTH

The thick wall vessel shown on Figure 2.36.1 was tested with an internal pressure, and also for thermal growth. Both tests were done using the 2-D and the 3-D continuum elements. Figure 2.36.1 shows the 2-D model. The 3-D model is made of 3 one-inch layers of the 2-D model. Again, H3DMAP results are in close agreement with the classical solution.

Test	Classical Solution			H3DMAP 2-D Element		H3DMAP 3-D Element	
Internal pressure P = 10,000 psi	r(in)	σ_R (psi)	σ_{θ} (psi)	σ_R (psi)	σ_{θ} (psi)	σ_R (psi)	σ_{θ} (psi)
	1.125	-7201	13868	-7188	13960	-6939	13670
	1.375	-3719	10385	-3701	10420	-3542	10210
	1.625	-1715	8382	-1700	8393	-1592	8242
	1.875	-459	7125	-445	7128	-370	7009
	r(in)	displacement u(in)		u(in)		u(in)	
	1.00	6.78×10^{-4}		6.53×10^{-4}		6.49×10^{-4}	
	1.25	5.78×10^{-4}		5.50×10^{-4}		5.46×10^{-4}	
	1.50	5.19×10^{-4}		4.86×10^{-4}		4.83×10^{-4}	
	1.75	4.82×10^{-4}		4.44×10^{-4}		4.41×10^{-4}	
	2.00	4.60×10^{-4}		4.17×10^{-4}		4.14×10^{-4}	
Thermal growth $\Delta T = 100^{\circ}\text{F}$	r(in)	u(in)		u(in)		u(in)	
	1.00	8.25×10^{-4}		8.45×10^{-4}		8.45×10^{-4}	
	1.25	1.06×10^{-3}		1.06×10^{-3}		1.06×10^{-3}	
	1.50	1.27×10^{-3}		1.27×10^{-3}		1.27×10^{-3}	
	1.75	1.48×10^{-3}		1.48×10^{-3}		1.48×10^{-3}	
	2.00	1.69×10^{-3}		1.69×10^{-3}		1.69×10^{-3}	

2.37 Case 37 - Stress Concentration - Hole in a Flat Plate

STRESS_CON

A flat plate with a hole in the center was modelled (Figure 2.37.1) using 2-D continuum elements. The plate was under a nominal load of 100 psi in the region far from the hole. The stress at the surface of the hole was obtained by extrapolating the stresses given in the middle of the elements. The stress concentration factor was then calculated and shown to be within the range of normally accepted values for this type of problem. For example, using the correlation for stress concentration factor given in Reference 58, the peak stress given at node 1 is 325 psi. The computer code prediction using interpolation to the inside radius gives 326 psi.

2.38 Case 38 - Simply Supported Circular Disk - Natural Frequencies

DISK4

A flat circular plate, 20 inches in diameter and 1/2 inch thick, was modelled using an axisymmetric solution. The natural frequencies were calculated theoretically using expressions given in Roark/48/ and Seireg/41/ and compared with the H3DMAP results. Two different meshes were used (4x20 and 4x40) to obtain the natural frequencies using H3DMAP. It can be observed in the following table that the finer mesh yields more accurate results. The first three mode shapes calculated by H3DMAP (4x40 model) are shown in Figure 2.38.1 (note that only half of the plate is shown).

Natural Frequencies (Hz)

Roark	Seireg	H3DMAP (4x20)	H3DMAP (4x40)
$f_1 = 487$	$f_1 = 502$	$f_1 = 487$	504
$f_2 = 1903$	$f_1 = 1957$	$f_1 = 1868$	1938
	$f_3 = 4375$	$f_1 = 4101$	4253

2.39 Case 39 - Simply Supported Beam - Natural Frequencies

BEAM_VIB

The 2-D continuum element (plane stress) was used to find the natural frequencies of the simply supported beam with a square cross-section as shown in Figure 2.39.1. Timoshenko/49/ has developed a solution taking into account the effect of rotary and shear deformations.

$$P_i = \frac{a\pi^2}{\lambda_i^2} \left[1 - \frac{1}{2} \frac{\pi^2 r_s^2}{\lambda_i^2} \left(1 + \frac{E}{k'G} \right) \right]$$

where: p_i is the i^{th} natural frequency

$$a = \sqrt{\frac{EI}{\rho A}}$$

$$\lambda_i = \frac{l}{i} = \text{length of the half waves into which the bar is subdivided during vibration}$$

$$k' = 0.833$$

$$r_t^2 = \frac{I}{A}$$

The following table compares the H3DMAP results, using compatible and incompatible modes, with the classical solution. Fig. 2.39.1 also shows the first three modes of deformation.

Natural Frequencies (Hz)

Mode of Vibration	Classical Solution	H3DMAP Compatible Mode	H3DMAP Incompatible Mode
1	447	468	446
2	1767	1804	1720
3	3893	3551	3400

2.40 Case 40 - Axisymmetric Buckling of a Circular Simply-Supported Plate

CIRPLATE_BUCK_IN

A flat circular plate, with the edge simply supported is loaded radially as shown in Figure 2.40.1. An axisymmetric model was used to analyze the plate. One hundred 2-D continuum elements were used in the x direction and four elements in the y direction.

The total applied load was 4,000 lbs. Theoretically, the critical load for this plate should be 29,000 lb. resulting in an eigenvalue of $\lambda=7.25$. For the compatible mode, H3DMAP yielded an eigenvalue of $\lambda=7.93$, while the incompatible mode gave $\lambda=7.245$.

2.41 Case 41 - Impulsively Loaded Beam

XBEAM1

In this example, a doubly built-in beam is subjected to an explosive load (ignition of a sheet explosive) at mid-span as shown on Figure 2.41.1. The finite element model uses an initial velocity to represent the energy

provided by the explosion. The H3DMAP results are shown in Figure 2.41.2, along with the experimental results/50/. For this class of problem, the agreement is quite good.

2.42 Case 42 - Axial Impulsively-Loaded Beam

XBEAMAX1

A constant load of 10 lbs. is suddenly applied axially to a cantilever beam. The model and the beam response are shown in Figure 2.42.1. Close agreement with the classical solution has been achieved.

2.43 Case 43 - Geometric Non-Linear Beam Test

XBEAM3

The test case used to verify the 3-D continuum element explicit solution (Case 56: SOLDEX7B) was modified to test the beam element explicit solution. The pressure (2.85 psi) was applied using point loads. The model and results are shown in Figures 2.43.1 and 2.43.2. The H3DMAP results compare well with the numerical solution obtained by T. Belytschko et al/52/. This test case was also solved using the implicit method and the same results were obtained. This test case was repeated using the explicit method with the shell element and solid elements respectively (Case 45: XSHELL1 and XSHELL1A, Case 56: SOLDEX7B).

2.44 Case 44 - Moment Load Test for Non-Linear Beam (Dynamic Relaxation)

XBEAM4

This test consists of bending an initially straight beam into a circular shape. The moment, M , required to bend the beam of length, L , into a circular shape is given by the curvature formula;

$$\frac{M}{EI} = \frac{\frac{d^2w}{dx^2}}{\left[1 + \left(\frac{dw}{dx}\right)^2\right]^{\frac{3}{2}}} = \frac{1}{R}$$

Thus,

$$M = \frac{EI}{R} = \frac{2\pi EI}{L}$$

An accurate solution will lead to a fully circular shape when M is applied. Since the static solution is sought, the moment M , required to bend the beam into a circular shape is applied as a ramp function (figure 2.44.1) and the problem is solved using dynamic relaxation.

Figure 2.44.1 shows the x and y displacements of the end node while Figures 2.44.2 and 2.44.3 are geometry plots of the beam at two different times in the analysis.

2.45 Case 45 - Linear and Geometric Non-Linear Shell Test

XSHEL1 and XSHEL1A

A cantilever beam shown in Figure 2.45.1 is used to test the linear and geometric non-linear response of the shell element. A uniform pressure is applied as a step function on the beam in the positive y-direction.

The linear response is obtained by applying a pressure of $P=0.01$ psi. Theoretically the maximum deflection should be twice the value of the static deflection. The first period is found using the following relationship obtained by Timoshenko, Young and Weaver/49/.

$$\delta_{max} = 2 \cdot \delta_{static} = 2 \cdot \left(\frac{wL^4}{8EI} \right)$$

$$T_1 = \frac{2\pi}{3.515} \sqrt{\frac{\rho AL^4}{EI}}$$

The computed and theoretical results compare very well as shown in the following table.

	Theory	H3DMAP
Maximum deflection δ_{max} (in)	0.0250	0.0246
Period T (sec)	0.00572	0.0058

A pressure of $P=2.85$ psi is used to verify the geometric non-linear response. This pressure amplitude results in large displacement response which is captured by the non-linear geometric shell formulation. Figure 2.45.1 shows the response of Node 6 and 12 in the y- direction and Figure 2.45.2 shows the deformed geometry at 4 ms. The deflection of Node 6 and 12 are in close agreement with Belytschko et al. results/52/. This test case was also done using the beam element and the solid element (Case 45: XBEAM3, Case 56: SOLDEX7B) where similar results were obtained.

2.46 Case 46 - Simply Supported Square Plate Test

XSHEL2 (elastic)

XSHEL2A (elastic-plastic)

A simply supported square plate is loaded with a constant downward pressure of 300 psi which is applied as a step function. One quarter of the plate was modelled as shown on Figure 2.46.1. Two material types were studied: elastic and elastic-perfectly plastic. H3DMAP results are shown on Figures 2.46.2 and 2.46.3 and compare very well with the results obtained by Belytschko et al/52/. A plot of the plate's response at time=0.7 ms is shown in Figure 2.46.4.

2.47 Case 47 - Impulsively Loaded Cylindrical Panel

XSHEL4A

As part of the shell element/61/ verification program, the H3DMAP predictions were compared against the experimental results obtained by Morino, Leech and Witmer/51/.

In this case, a cylindrical panel is impulsively loaded with an explosive as shown in Figure 2.47.1. The panel is clamped along the side edges and simply supported at both ends. The explosion is simulated by an initial velocity which yields the same impulse obtained by the explosive.

In the experiment/51/, the boundary of the panel in contact with the explosive is partially constrained against rotation. This requires special attention since this condition lies between the ideal boundary constraints of simply supported and fully fixed. To circumvent this problem, the simple support was modelled using a plastic hinge, i.e. using a softer material in the boundary region, and the hinge is fully restrained as far as the computer code is concerned. Consequently, the first layer of elements rotates more easily than the rest of the panel while, at the same time, offering some resistance to rotation which is not provided with normal simple supports. This approach provides a more realistic representation of the boundaries present in the experiment.

Figures 2.47.2 to 2.47.4 compare experimental data and H3DMAP results while Figure 2.47.5 shows the deformed structure after 0.8 ms. Fig. 2.47.6 and 2.47.7 show the plastic strain in the structure.

The differences observed between the experimental and computed results can be attributed to both the difficulty in accurately modelling the boundary conditions of the experiment and the discretization error. However, the agreement in results is very good for this class of problems.

2.48 Case 48 - Impulse Load on a Circular Plate

XSHELS

One of the verification cases used to validate H3DMAP Version 4 was taken from Reference 53. This case involved the experimental evaluation of the response of impulsively loaded clamped circular plates. It is similar to the type of analysis being performed on the transportation cask designs where an initial velocity is applied to a structure. The model geometry used for both the experiment and analysis is shown in Figure 2.48.1. The results reported herein correspond to test P-8 in Reference 53 for an initial velocity of 4410.0 in/sec. The material is Aluminum 6061-T6 and the properties are given in Figure 2.48.1. The analysis was carried out using shell elements in order to verify the validity of the element explicit formulation. The shell model is shown in Figures 2.48.2 and 2.48.3.

The comparison of the predicted vertical central deflection (node 1) time history with the experimental value is shown in Figure 2.48.4. Although some error is observed, Reference 53 indicates that slippage occurred between the clamping surface and the plate after detonation and the effective radius increased slightly, leading to somewhat higher responses. The correlation is excellent for this class of problem. A typical plot of the deformed structure is shown in Figure 2.48.5.

2.49 Case 49 - Moment Load Test for Non-Linear Shell (Dynamic Relaxation)

XMOMSHEL

The purpose of this test case was to verify the capacity of the shell element formulation to handle geometric non-linear deformation. This case is similar to Case 44 only using shell as opposed to beam elements. The test consists of bending a strip of initially straight shell into a circular shape. The model is shown in Figure 2.49.1

along with the material properties and the applied load. Fig. 2.49.2 shows the displacement time history of node 13 in both x and y direction while Figures 2.49.3 and 2.49.4 depict the response of the metal strip during and at the end of loading, respectively. From this example, it can be seen that the shell element formulation easily handles this type of large deformation problem.

2.50 Case 50 - Corner Supported Flat Plate Under Pressure Load (Explicit Solution)

XCORNSHEL

This is the same as Case 32 except that now it is solved using the explicit solution. Again, a quarter mesh model is used as shown in Figure 2.50.1. Note that this mesh is finer than the one used for the implicit case. Boundary node 9 is constrained in the translational degrees of freedom. The corner point loading of the plate in this case provides an excitation mechanism for the w- hourglass mode described in Reference 57. Inadequate control of the hourglass mode leads to unacceptable mesh distortion. Hourglass control is obtained using the viscous form of control described in Section 3.9 of Reference 57 using an hourglass stabilization coefficient of 0.1. The purpose of this test was to verify the hourglass control algorithm. The results obtained in this case are in good agreement with the classical solution and the results obtained by H3DMAP using the implicit solution scheme (Figures 2.50.2 and 2.50.3).

2.51 Case 51 - Axial Impact of a 3-D Beam - Rigid Wall

SOLDEX1

A 3-D straight beam (Figure 2.51.1) was modelled. The $x=0$ plane impacted against a rigid wall positioned at $x = -0.002$ in. The initial velocity of the beam was 150 in/sec in the negative x-direction. The boundary conditions of each node were fixed in all degrees of freedom with the exception of the x-direction. The numerical solution agrees with the classical solution as can be seen in Figure 2.51.1. This test case also verified the rigid wall algorithm.

2.52 Case 52 - Axial Impact of a 3-D Beam - Boundary Restrictions

SOLDEX2

This test case is almost identical to the previous one with the exception of the rigid wall being removed and replaced by fixing the boundary conditions in the x-direction for all the nodes in the $x=0$ plane. Fig. 2.52.1 shows the results of this test case.

2.53 Case 53 - Simply Supported Beam - Step Load at Mid-Span

SOLDEX3

This test case verifies the transverse response of a simply supported beam to a concentrated force applied at mid-span as shown on Figure 2.53.1. The boundary conditions were fixed for all nodes in the y-direction to prevent lateral movement. H3DMAP results are shown in Figure 2.53.2 and it can be observed that the results are in close agreement with theory. In addition, the same test case was run using the implicit operator, and similar results were obtained.

2.54 Case 54 - Axial End Load on Cantilever Beam

SOLDEX5

In this test case, a 3-D cantilever beam, Figure 2.54.1, is axially loaded at its free end in the manner shown by the pressure vs. time curve (see Figure 2.54.2). Results are also shown and compared with theory where good agreement is observed.

2.55 Case 55 - Dynamic Relaxation

SOLDEX6 - SOLDEX 6A

The same model described in the previous test case was used to test dynamic relaxation. Again, pressure was applied at the free end of the beam. Fig. 2.55.1 shows the pressure vs. time loading curve and the material elastic-plastic properties. The H3DMAP results are shown in Figures 2.55.1 and 2.55.2.

This test case has been modified to test dynamic relaxation of a purely elastic material. A new load curve had to be used. Figs. 2.55.3 and 2.55.4 show H3DMAP results along with load-curves used and the classical solution.

2.56 Case 56 - Geometric Non-Linear Beam Test

SOLDEX7B

A cantilever beam is subjected to a constant lateral pressure as shown in Figure 2.56.1. The degree of freedom in the z direction has been restricted for all nodes to prevent any undesired lateral movements. Figures 2.56.2 to 2.56.3 show H3DMAP results which were found to be in good agreement with the numerical solution obtained in /52/. The same test case has been solved using beam elements and shell elements (Case 43: XBEAM5, Case 45: XSHEL1 and XSHEL1A).

2.57 Case 57 - Undamped Spring-Mass System with Initial Velocity

XSPRING

The spring-mass system shown on Figure 2.57.1 is excited by an initial velocity of 5 in/sec. The displacement and velocity response are shown in Figures 2.57.1 and 2.57.2 respectively. The predicted values, obtained by numerically integrating the equations of motion using the central difference operator method, are shown in Figures 2.57.3 and 2.57.4. The results are in excellent agreement with classical results.

2.58 Case 58 - Undamped Spring-Mass System with Force Step Function

XSPRINGF

A constant amplitude force was applied to a spring-mass system as shown in Figure 2.58.1. Comparing with H3DMAP results, it can be seen that the numerical and theoretical solutions are in good agreement.

2.59 Case 59 - Undamped Spring-Mass System - Beat Response

XBEAT_SPRING

The same spring-mass system depicted in the previous test case is excited by a periodic force $f(t) = F_0 \sin \omega t$. The beat phenomenon occurs when ϵ gets small in the following equation $|\omega - \omega_n| = 2\epsilon$, where ω_n is the natural

frequency of the system and w is the forcing frequency. The beating period is equal to π/ϵ , which in this case is $T_p = 6.28$ sec for $w_s = 10$ rad/s and $w = 11$ rad/s. This theoretical value compares well with the H3DMAP result as shown on Figure 2.59.1.

2.60 Case 60 - Annular Element

XANNULUS

Node 1 of a spring element is trapped inside an annular element, and is subjected to the action of two forcing functions as depicted in Figure 2.60.1. The trajectory of Node 1 as it moves around and bounces off the annular wall is also represented on the same figure. When the total time of the run is increased, the whirling motion, characteristic of this type of excitation, can be observed. This test was also done using the implicit solution in Case 11 (MAP13).

2.61 Case 61 - Sandwich Model - Dynamic Relaxation

XGAPTEST1 and XGAPTEST1A

Figure 2.61.1 shows a kind of sandwich model consisting of three shells interconnected with gap or spring elements. A ramped force was applied on one side of the sandwich while movement of the other side was restricted. The dynamic relaxation option was used. Results are shown in Figures 2.61.1 and 2.61.2 and are in excellent agreement with the theory.

2.62 Case 62 - Fluid-Solid Interaction of Tritiated Heavy Water Transportation Container

FLUIDTEST3

A numerical simulation/59/ of the tritiated heavy water transportation container package (THWTP) undergoing an accidental impact represents an interesting challenge to test a variety of features offered in H3DMAP, especially the hydrodynamic material formulation. The overall model for the 9m flat-end drop simulation is shown in figure 2.62.1. The multi-material structure where steel and polyurethane foam interface account for one of the complexities of the problem. In addition, consideration of the fluid-structure interaction of a liquid payload with a containing vessel adds another difficulty. The explicit solution model (LTYP=8) was used to simulate the drop. Since both the THWTP and the drop orientation are symmetric, one quarter of the package is modelled with appropriate boundary conditions on the planes of symmetry. The THWTP assembly is modelled using several component models which represent: a) the overpack outer and inner shells and foam core, b) D₂O container, c) liquid payload and d) the overpack upper lid portion. The total weight of the assembly is 35,600 lbs. The mesh provided the refinement needed in the vicinity of the point of initial impact while becoming progressively coarser in areas remote from the impacting surface. The model consists of 1791 nodes with 1254 solid elements representing the polyurethane foam, fluid, and rigid body portion of the model. In addition, 564 shell elements/57,61/ represent the stainless steel D₂O inner container and the inner/outer liners of the overpack. The analytical model is shown in figures 2.62.2, 2.62.3 and 2.62.3a. The clearance between the inner D₂O container and the inner liner of the overpack is accounted for with interface elements. The inner surface of the overpack defines the surface which limits the potential motion of the inner container. The target is modelled as a rigid unyielding surface.

The results indicate considerable interaction of the liquid payload, the inner container, and the inner liner of the overpack (figure 2.62.4). The side of the D₂O container and the inner liner of the overpack first come into contact at 2.6 ms. The fluid initially responds acoustically until the first shock wave reaches the D₂O container wall at 0.6 ms. At this point the fluid response is attenuated by the flexibility of the container shell and subsequently by the combined effects of the container and the overpack when they impact later in the transient. Essentially the fluid response is governed by the deceleration. The distribution in the hydrostatic pressure of

the fluid is mainly due to the variation of flexibility of the inner container as it interacts with the overpack. The time history of hydrostatic pressure at the bottom of the inner container in the centre and the edge is shown in figure 2.62.5. Figures 2.62.7 to 2.62.12 show the hydrostatic stress distribution at different times during the impact. Figures 2.62.13 and 2.62.14 show the effective plastic strain deformation. A maximum permanent increase in the diameter of the inner D.O container of 1.28 inches occurs near the bottom where the fluid pressure is greatest.

The analysis was verified by a nine-metre drop test, using a quarter-scale model of the THWTP. This model included all of the main components of the transportation package, including the liquid payload container and foam-filled overpack. It was instrumented with two pressure transducers mounted at the bottom of the water container and four accelerometers: two on the bottom of the water container and two on the outer surface of the overpack. Figure 2.62.6 shows the pressure/time histories from the pressure transducers, one near the centre of the container and one close to the edge. It should be noted that the time span of the test result is compressed relative to the analysis because of geometric scaling effects. The maximum permanent increase in diameter of the 1/4 scale experimental model was measured at 0.30 inches which corresponds to 1.20 inches full scale. Considering the complexity of the problem, the shifting of the D.O container in the clearance during the drop, and the difficulty in ensuring a perfect experimental 9m flat-end drop, excellent agreement between analytical predictions and test have been obtained as observed in figures 5.62.5 and 5.62.6.

2.63 Case 63 - Fluid Hydrodynamic Material Model Testing Equations of State Type 1 and 2

HYDRODYN1, HYDRODYN2

Consider a 1-D rigid pipe, filled with water and travelling at an initial velocity $V_i = 100$ in/s, hitting a rigid surface. This is similar to the case of an instantaneous closing of a valve resulting in waterhammer (see Figure 2.63.1). At impact, a pressure wave travels back and forth in the water column.

Using the following properties:

Total Length	= $L_i = 1,000$ in.
Density of Water	= $\rho = 0.0361$ lb _m /in ³
Bulk Modulus	= $K = 300,000$ psi

the velocity of the pressure wave, the position of the pressure wave, the dynamic component of the pressure and the maximum displacement of the free end of the column opposite to the impact can be obtained analytically.

$$\text{Acoustic Velocity} = C = \sqrt{386.4 \frac{K}{\rho}} = 56,745 \text{ in/sec}$$

At $t = 0.00580$ sec., the wave is located at 329 in. from the fixed end and at $t = 0.01077$ sec., the wave is at 611 in. from the fixed end.

$$P_d = \rho C V_i = \frac{0.036}{386.4} \times 56745 \times 100 = 529 \text{ psi}$$

$$\delta_v = \frac{L \Delta V}{V} = \frac{P_d L}{K} = \frac{529}{300,000} \times 1000 = 1.763 \text{ in.}$$

As seen from Figures 2.63.2 and 2.63.3, the shock wave propagation obtained from the H3DMAP code agrees closely with both the amplitude and shape of the theoretical solution. The pressure jump across the wave front is not as sharp as in theory due to the finite size of the elements and the use of artificial bulk viscosity which smears the shock over a number of elements. In Figure 2.63.4, excellent agreement is observed in the time history displacement of the free surface. Both equation of state Type 1 and Type 2 have been tested, and the results were identical.

2.64 Case 64 - Dynamic Relaxation Test - 3-D Cantilever Beam

SOLDEX6A

A simple cantilever beam model with a pressure load applied on the free end as shown on Figure 2.64.1 has been used to test the dynamic relaxation option on H3DMAP. The pressure is applied using a steep ramp. The displacement response of the free end along with the stress response in the element adjacent to the free end are plotted in Figures 2.64.2 and 2.64.3 respectively. Numerical and theoretical results are in excellent agreement in this case.

2.65 Case 65 - Pipe Whip Test

WHIP

The following test case was designed to show the capability of H3DMAP to handle pipe whip problems. The pipe shown in Figure 2.65.1 was rotated around a pivot in such a way that the free end of the pipe would hit a rigid wall at an angle of 30° and a velocity of 1250 in/sec. The deformed structure at time = 3.5 ms of the pipe is shown in Figure 2.65.2. The corresponding effective plastic strains and Von Mises stresses are shown in Figures 2.65.3 to 2.65.6, and the X-displacement time history of node 1 is shown in Figure 2.65.7.

2.66 Case 66 - Tritiated D.O Transport Container Punch Test

PUNCH

Mathematical Model

In order to demonstrate the validity of the analytical models used for shell and solid modelling in H3DMAP and provide some insight into the error bounds which can be expected, a benchmark with an experimental model was performed. Details are given in References 54 and 55. The comparison serves to test the polyurethane foam and stainless steel material models and demonstrate the use of the shell elements in combination with solid elements representing hydrodynamic materials. The model used an outer shell thickness of 0.2920 inches in order to provide for a failure of the outer shell in punching shear. This would enable a benchmark of the failure limits used in the analysis of radioactive waste transportation containers in order to confirm that they were both conservative and realistic.

The case considered, was the accident scenario involving a 4 ft drop (impact velocity: 192.6 in/sec) of the package analyzed in Reference 55 onto a mild steel pin. The material properties used are given in Reference 55. The yield versus pressure curve and the equation of state defining the material model for the polyurethane foam are given in Figures 2.66.4 and 2.66.5 respectively. Due to the localized nature of this drop, only the lower portion of the overpack was modelled as shown in Figure 2.66.1. It consisted of polyurethane foam sandwiched between a 0.2920 inch thick outer plate and a 0.2692 inch thick inner plate.

The model topology is shown in Figures 2.66.1, 2.66.2 and 2.66.3. There are a total of 1574 nodes, 1140 solid elements representing the foam and 416 shell elements for the plates. The pin was modelled using the partial rigid wall option in H3DMAP.

Since the model only partially represents the package weight, the remaining weight of package plus contents was included as a non-structural mass. The total weight of the package is 31,550 lbs. This includes the inner container with payload and the overpack. Since the experimental model was ballasted in such fashion that the remaining package weight was uniformly distributed across the 0.2692 in inner liner plate, the same approach was adopted here by increasing the density of the inner liner plate.

Experimental Model - 1/4 Scale

The experimental model is described in detail in Reference 54. It is a 1/4 scale model of the bottom portion of the overpack. The model was instrumented to obtain impact force and displacement time histories. The drop height was 4 ft and the experimental model was ballasted to simulate the total package weight.

Scaling Laws

In order to compare the results of analysis and experiment the following relationships between scale model (m) and full scale prototype (p) were used:

Deflection	Y_m	=	Y_p/n
Stress	S_m	=	S_p
Force	F_m	=	F_p/n^2
Energy	E_m	=	E_p/n^3
Mass	M_m	=	M_p/n^3
Time	T_m	=	T_p/n
Acceleration	A_m	=	nA_p

where $n = 4$ for 1/4 scale model.

These relations were applied to the experimental results and the comparison with analysis presented with reference to full scale.

Analytical Results and Comparisons

The impact simulation was run to 36 ms. In the experiment, failure of the outer shell occurred at approximately 28 ms in the form of a punch through. A deformed plot at that time is shown in Figure 2.66.6. The comparisons with experimental results are shown, in terms of the full scale quantities of displacement and impact force time histories in Figures 2.66.7 and 2.66.8. The details of the experimental results are given in Reference 54. As demonstrated in the plots, excellent agreement is obtained for impact forces and displacements. At failure time (28 ms), a total peak impact force in the analytical model of 460,000 lbs. is observed.

Figure 2.66.9 shows the plastic strain as a function of time in the shell element underneath the edge of the pin, while Figure 2.66.10 describes the hydrostatic stress evaluation in the foam element directly below the center of the pin (refer to Figure 2.66.3 for the location of these elements).

This benchmark serves to demonstrate the accuracy of the mathematical modelling through the excellent correlation obtained between the experimental and analytical results.

Plastic strains in the shell elements at the time of failure are shown in Figures 2.66.11 and 2.66.12. The corresponding von Mises stresses and hydrodynamic stresses in the foam elements are given in Figures 2.66.13 and 2.66.14.

2.67 Case 67 - Corner Drop Model

XCONE

In Reference 56, a simplified method has been developed for predicting the corner-drop impact loads of solid-walled transportation packages for radioactive materials. The method is based on the observation that a corner drop impact is a highly localized phenomenon and that almost the entire impact energy is dissipated by the plastic deformation of the material in the impact zone. It has been shown that the corner drop impact of a transportation package is equivalent to the impact of a cone hitting an unyielding target, where the cone represents the corner of the package, and an equivalent weight at the top of the cone represents the weight of the rest of the package (see Figure 2.67.1). For this simple geometry, structural evaluation of the cone can be carried out using classical strength of material methods.

The analytical results presented in Reference 56 of the cone impact have been compared with H3DMAP predictions. A quarter model of the cone was constructed as shown in Figure 2.67.2, using 3-D continuum elements. Fig. 2.67.3 shows a plot of the impact force vs. time as obtained by H3DMAP (results of the quarter model multiplied by 4). The results are in excellent agreement with the analytical value of the maximum impact force $F = 19.8 \times 10^6$ lbs.

Fig. 2.67.4 depicts the deformed shape of the structure while Figure 2.67.5 shows the distribution of effective plastic strain in the cone.

2.68 Case 68 - Sphere Impact Analysis

Experimental results for a 0.6096m (24") diameter, 0.01588m (5/8") thick, Type 304 stainless steel, sphere impacting a concrete containment is given in reference /62/. Figure 2.68.1 describes the problem and gives the problem dimensions, material properties, and the impact velocity. The barrier is assumed to be rigid. The sphere is modelled as shown in Figure 2.68.2. The model consists of 222 8-noded hexahedron elements. Symmetric boundary conditions are imposed at the free edges.

The analysis indicates that the duration of impact is 3 milliseconds, after which the sphere rebounds. The deformed shape of a section of the sphere at the end of the analysis is as shown in Figure 2.68.3. Figure 2.68.4 gives a comparison between H3DMAP and experiment for various dimensions of the deformed shape. Figure 2.68.5 gives a time history plot of the diametrical reduction obtained using H3DMAP and experiment. Figures 2.68.4 and 2.68.5 indicate excellent agreement between H3DMAP and experimental results.

2.69 Case 69 - Radioactive Filter Transportation Package - 9m Corner Drop Impact

General Description

In Reference 64, an experimental 9m drop of a 1/4 scale model of the Radioactive Filter Transportation Package (RFTP) onto an unyielding target was carried out. The RFTP was dropped on the bottom corner with the centre of gravity directly over the impact point as shown in figure 2.69.5. This is the most severe orientation and the damage on the experimental model can be readily observed in figures 2.69.6 to 2.69.8.

The 1/4 scale package, shown in Figure 2.69.1, comprises a payload, foam dunnage, and a polyurethane foam-filled overpack. The overpack is a sandwich construction with 304L stainless steel liners and 3.125 inch nominal foam walls. The outer shell is 0.1875 inch thickness and the inner shell is 0.0625 in. The primary containment boundary consists of the overpack inner housing, the containment lid and outer containment seals in the lid region.

Mathematical Model

The purpose of this work is to benchmark the explicit module of the computer code H3DMAP. Figure 2.69.2 shows the mathematical model and the drop configuration with the centre of gravity over the impact point. The package assembly is modelled in several components which are shown in Figure 2.69.3. Symmetry about a vertical plane permits the use of a half model. Due to the highly localized nature of this accident scenario, mesh refinement was confined to the vicinity of initial impact as observed on the template that was used to model the RFTP (figure 2.69.4).

The overpack body, consisting of inner and outer stainless steel shells with polyurethane foam between the shells, is modelled using four-noded thin shell elements for the steel and eight-noded continuum elements for the foam. The package contents, namely the payload and the foam dunnage, are modelled using continuum elements. The shell and continuum elements share common nodes where they join. The model consists of 4786 nodes, 1428 shell elements, and 3740 solid elements. The calculated total weight of the final model is 487.8 lbs compared with an actual total weight of 486.5 lbs. A master-slave contact surface is defined between the inside of the overpack and the dunnage-payload assembly. In the experiment, the impact target for the drop test is a 4 in thick steel plate on a solid concrete foundation. It has been modelled using the rigid wall option in H3DMAP.

The stainless steel material was modelled using an isotropic elastic-plastic constitutive law. The principal axes/65/ technique was used to account for large material rotations (Ref. 61). Strain rate insensitive behaviour for the 304 L SS material is assumed. The foam material was modelled using a hydrodynamic elastic-plastic model with an equation of state.

The loading on the models was obtained in terms of an initial velocity applied to the complete region defining the package, using the following relation from rigid body mechanics:

$$V_o = [2gh]^{1/2}$$

where: V_o = initial velocity at impact (in/sec)

h = drop height (in)

g = gravitational constant (386.4 in/sec/sec)

for h = 29.53 ft (9 m) V_o = 523.3 in/sec

Analytical Results

The 9m edge drop simulation was run to 8 ms. The initial drop energy and momentum were dissipated at approximately 6 ms. The localized nature of this problem is readily observed in the deformed structural plots of Figures 2.69.9 to 2.69.20. The post impact deformation of the experimental model is shown in Figure 2.69.8. Comparison with the predicted deformation shown in Figures 2.69.17 and 2.69.20 as well as comparison of the experimental profile and the predicted one in Figure 2.69.32 indicate excellent agreement. Figures 2.69.21 to 2.69.26 show the evolution of the hydrostatic stress distribution as the impact progresses. The effective plastic strain in the shells after the impact can be observed in Figures 2.69.27 to 2.69.30.

The impact force time history on Figure 2.69.31 indicates a peak impact force of 86,000 lbs for the half model in the drop direction Z. This corresponds to a maximum deceleration $\ddot{Z}_m$ of the total package of:

$$\ddot{Z}_m = \frac{86000}{243} = 354g$$

In the experiment, the peak deceleration for the package was 348 g. This again shows excellent agreement between the analytical and experimental results.

2.70 Case 70 - Rolled Joint Simulation

Mathematical Model

In reference /63/, a study was undertaken by Ontario Hydro for the CANDU Owners Group (COG) to develop a capability for proper analytical modelling of the pressure tube rolling process (Figure 2.70.1). A numerical model for roller kinematics was incorporated into the non-linear transient explicit module of H3DMAP in order to provide a comprehensive simulation model for the rolling process. This case serves to demonstrate the capability of the three-dimensional contact algorithm as well as the roller kinematic option. The most important aspect of the finite element modelling of the rolling process is the correct application of loads that cause the rolling. The model used here is made up of three separate parts: pressure tube, end fitting, and roller assembly. All of these parts exert forces on each other through sliding interface contact surfaces. Three dimensional linear isoparametric hexahedron elements with reduced integration are used to model the continuum.

The pressure tube has three elements through the thickness, 45 elements around the circumference at 8° spacing, and twelve elements in the axial direction. Similarly, the end fitting has three elements through the thickness and 45 elements circumferentially, but only five elements in the axial direction. One row of elements has been removed, and some nodes have been adjusted to more accurately reflect the shape of an actual end fitting. The pressure tube and end fitting elements are shown in Figure 2.70.2. The five rollers are made up of 256 elements each with eight layers of elements in the axial direction. The roller mesh is refined in the region of contact with the inside of the pressure tube, and the rollers are evenly spaced at 72° circumferentially. Figure 2.70.2 shows a cross-sectional view and end view assembly with relevant dimensions. A total number of 5535 nodes and 3530 solid elements were used. Two views of the complete model in the initial configuration are given in Figure 2.70.3. The three parts of the model are connected by contact surfaces which are specified in pairs. The first set consists of the pressure tube outside surface and the end fitting inside surface. The second set is made up of the pressure tube inside surface and the roller outside surfaces. Both sets of contact surfaces are shown in Figures 2.70.4 and 2.70.5. A colour coded figure displaying the assembled finite element model is given in Figure 2.70.7.

The material properties used in the finite element model are given as follows;

	Pressure Tube	End Fitting	Rollers
Young's Modulus, E	13.7×10^6 psi	29.2×10^6 psi	50×10^6 psi
Poisson's Ratio	0.38	0.283	0.3
Yield Strength	74000 psi	85000 psi	--
Work Hardening Slope	0.2×10^6 psi	50000 psi	--

The pressure tube is restrained axially at the end opposite to the end being rolled and the roller cage is restrained from axial movement relative to the outboard end of the end fitting. Friction keeps the pressure tube from slipping in the end fitting. A rotating tapered mandrel is drawn into the roller set by action of the rollers being skewed, and the rollers are thus forced radially outwards as they revolve. The force exerted on the pressure tube by the rollers results in the application of hoop and bending stresses as well as compressive rolling

forces on the pressure tube. Thinning of the pressure tube causes some extrusion which changes the axial position of the free end of the pressure tube and end fitting.

In order to have a relatively simple model, boundary conditions are applied in a slightly different way from the actual rolling process. To avoid the need to model friction between the pressure tube and end fitting, the end of the pressure tube opposite to the end fitting is fixed from rotating but is allowed to extend axially. The rolled end of the pressure tube and end fitting are restrained axially but are allowed to rotate and expand. The motion of the rollers is specified as a rotation through 90° about the centre of the roll set combined with an outward radial motion. This motion is followed by an inward radial retraction of the rollers to end the rolling process. The boundary conditions and specified roller motion effectively model:

- 1) combined rolling and outward radial motion of rollers
- 2) application of rolling pressure to pressure tube
- 3) twisting action of pressure tube and end fitting due to rolling
- 4) elongation of pressure tube due to rolling
- 5) removal of roll set to obtain residual stress in pressure tube and end fitting.

While the rollers undergo several revolutions of rotation in reality, here only 90° is used to simplify the modelling, and demonstrate the modelling principles involved.

Analytical Results

The simulation of the rolling process ran to completion as modelled. The progression of roller motion and deformation from the initial to final configuration is shown in Figures 2.70.6a to 2.70.6d. Deformation of the pressure tube is not prominent but some rotation relative to the end fitting is noticeable.

The distribution of effective plastic strain in the entire model, and in the pressure tube alone is shown in Figures 2.70.8 and 2.70.9, respectively. Some negligible plasticity is evident in the end fitting because:

- 1) the outward radial motion of the rollers is excessive (for demonstration purposes).
- 2) the actual complete end fitting is more rigid than the portion used in the model.

Modelling of more realistic rolling conditions is not expected to cause yielding of the end fitting. In Figure 2.70.9, the plastic strain is greatest near the inboard end of the rollers (transition region) because of the combined effect of rolling pressure and torsion. Torsion in the pressure tube is caused by the manner in which the pressure tube is restrained from rotating in the model. In reality, the rolling torque is transmitted to the end fitting by friction between the pressure tube and end fitting. There is visible twist in the pressure tube in the transition region. Also, plastic strain is greater on the outside surface of the pressure tube due to the combination of tensile bending stress (the pressure tube bends into a lobular shape under the local roller forces) and rolling pressure. Near the inboard end of the rollers, the pressure tube is in a very complex state of stress (rolling pressure + torsion + bending) during the rolling process. Plastic strain inboard from the rollers is the result of excessive torsion from the extreme roller motion and dynamic ringing of stress. This plastic strain should not occur with more realistic loading. Specific details of this work are given in reference 63.

2.71 Contact Algorithm Tests

Besides the 9m corner drop impact (Case 69) involving a radioactive waste transportation package and the rolled joint simulation (Case 70), a number of other tests were performed on the contact algorithm. A selected number are briefly described in this section.

Momentum Transfer

One of the fundamental tests performed on the contact algorithm/60/ developed for the explicit module of H3DMAP, was the one-dimensional impact of an elastic body against a rigid, fully constrained body. Three dimensional hexahedron elements were used to model the bodies as shown in Figure 2.71.1. A master surface was defined on the rigid body (body 2) and the corresponding slave surface was defined on the elastic body (body 1). An initial velocity of 100 in/s was applied to body 1. As seen in the time history of velocity at nodes 1, 2 and 3 in the x-direction on the interface, body 1 rebounds with the correct velocity.

Three Dimensional Indentor

The geometry for the case of a cylindrical indentor deforming a body is shown in Figure 2.71.2. The case consists of a rigid cylindrical body moving at a constant prescribed velocity into a deformable body with elastic-plastic properties. The symmetric treatment of master-slave surfaces are used. In this treatment, both the indentor and deformable body have defined master/slave surfaces. The evolution of contact between the two bodies is given in Figures 2.71.3 and 2.71.4.

Shell-Solid Contact

A simple case described in Figure 2.71.5 was used to test the contact algorithm for cases involving shell to solid contact. This case consisted of a plate impacting a rigid solid punch at an initial velocity of 523 in/s. The deformation at the point of maximum impact is shown in Figures 2.71.6 and 2.71.7.

Discontinuous Surface Contact

In order to test the contact algorithm for cases involving severely deforming surfaces which are discontinuous, two tests were set up. The geometry for both cases are shown in Figures 2.71.8 and 2.71.11. In each case, a projectile, defined using rigid solid hexahedron elements, and given an initial velocity impacts a shell structure. Both problems differ only in the trajectory of the projectile. The deformation for model 1 after the simulation is shown in Figures 2.71.9 and 2.71.10. For model 2, it is interesting to follow the deformation as it proceeds in time. The responses for model 2 at times 0.001, 0.002 and 0.006 sec. are given in Figures 2.71.12, 2.71.13 and 2.71.14, respectively. Given the extent of the deformation, the discontinuity of the channel shaped shell poses a severe test for the contact algorithm.

2.72 Finite Elastic Simple Shear

This test case serves to demonstrate the capabilities of the code in finite deformation using the principal axes algorithm/61,65/. As shown in figure 2.72.1, a shell element is subjected to large in-plane shear by imposing a specified constant velocity on the top horizontal surface. An isotropic elastic constitutive law is applied on the principal axes. As observed in figure 2.72.1, the results obtained are virtually indistinguishable from those obtained using the closed form finite elastic solution. This case effectively tests the principal axes algorithm.

ACKNOWLEDGMENTS

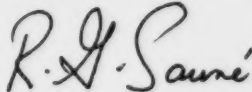
The authors would like to express appreciation to Mrs. C. Robshaw for her patience and skill in typing this report. Special thanks to Dr. T. Byrne for his useful comments.

Approved by:



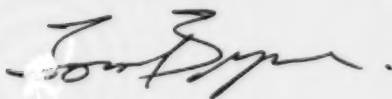
G.J. Clarke
Manager
Mechanical Research Department

Submitted by:

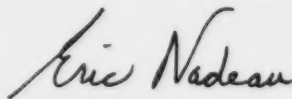


R.G. Sauvé
Unit Head-Applied Mechanics Development
Applied Mechanics Section

Reviewed by:



T.P. Byrne
Acting Section Head
Applied Mechanics Section



E. Nadeau
Engineer
Applied Mechanics Section

RGS/EN:cr

REFERENCES

1. Przemieniecki, J., *Theory of Matrix Structural Analysis*, McGraw-Hill, 1968.
2. Zienkiewicz, O., *The Finite Element Method*, McGraw-Hill, 1977.
3. Wilson and Bathe, *Numerical Methods in Finite Element Analysis*, Prentice Hall, 1976.
4. Hurty and Rubenstein, *Dynamics of Structures*, Prentice Hall, 1964.
5. Thomson, *Theory of Vibration with Applications*, Prentice Hall, 1972.
6. Biggs, *Introduction to Structural Dynamics*, McGraw-Hill, 1964.
7. Sauvé, R.G., *Applications of Eigensolution Techniques Using Mini Computers*, Proceedings Vol I, Fourth Symposium of Engineering Applications of Solid Mechanics, Toronto, 1978.
8. Paidoussis, M.P., *A Review of Flow-Induced Vibrations in Reactors and Reactor Components*, Proceedings Vol II, 2nd International Topical Meeting on Nuclear Reactor Thermal-Hydraulics, Santa Barbara, 1983.
9. Pettigrew, M.J., Gorman, D.J., *Experimental Studies on Flow-Induced Vibration to Support Steam Generator Design, Part III - Vibration of a Small Tube Bundle in Liquid and Two-Phase Cross Flow*, Paper No. 424, International Symposium on Vibration Problems in Industry, Keswick, U.K. (1973), also Atomic Energy of Canada Limited, Report 5804.
10. Connors, H.J., *Flow-Induced Vibration and Wear of Steam Generator Tubes*, Nuclear Technology, Vol 55, November 1981.
11. Pettigrew, M.J., Sylvestre, Y., Campagna, *Flow-Induced Vibration Analysis of Heat Exchanger and Steam Generator Designs*, AECL Report No. 5826, August 1977.
12. Sauvé, R.G., et al, *Seismic Qualification of Safety Related Instrumentation Cabinets for Nuclear Generating Stations: An Economical Approach*, Atomic Energy of Canada Ltd., Report No. 6548, June 1979.
13. CSA Standard N289.3, *Design Procedures for Seismic Qualification of CANDU Nuclear Power Plants*, April 1979.
14. Sauvé, R.G., Teper, W., *Non-Linear Dynamic-Impact Simulation of Process Equipment Tubes with Tube/Support Plate Interaction*, Proceedings Vol II, 2nd International Topical Meeting on Nuclear Reactor Thermal-Hydraulics, Santa Barbara, 1983 (also Ontario Hydro D&D Report No. 82156).
15. Sauvé, R.G., *Generation of Secondary Response Spectra - An Approach (Subroutine GENSPEC)*, Ontario Hydro D&D Report No. 82046, January 1982.

16. Cowper, G.R., *The Shear Coefficient in Timoshenko's Beam Theory*, ASME Journal of Applied Mechanics, June 1966.
17. Fogarty, J.P., Sauv , R.G., *Frequency Response of Pickering NGS B Primary Heat Transport Pump Test Loop*, Ontario Hydro D&D Report No. 83292, August 1983.
18. Peterson and Bathe, *Non-Linear Dynamic Analysis of Reactor Core Components*, Engineering Analysis Corporation, Report No. S- 104.3, Berkeley, California, 1972.
19. Wylie, R.C., *Advanced Engineering Mathematics*, 4th Edition, McGraw-Hill, 1975.
20. Youtsos, T.G., et al, *Elastic Analysis of Beam Support Impact*, American Society of Mechanical Engineers, Paper 82-PVP-58, 1982.
21. Hunt, K.H., Crossley, F.R.E., *Coefficient of Restitution Interpreted as Damping in Vibroimpact*, Journal of Applied Mechanics, ASME, June 1975.
22. Sauv , R.G., *Two-Dimensional Dynamic/Modal/Static Analysis Computer Program*, Ontario Hydro D&D Report No. 81191, April, 1980.
23. Collins, R.J., *Bandwidth Reduction by Automatic Renumbering*, International Journal for Numerical Methods in Engineering, Vol. 6, 1973.
24. Cuthill, E., and McKee, J., *Reducing the Bandwidth of Sparse Symmetric Matrices*, Proc. ACM Nat. Conf., 1969.
25. Fl gge, W. et al, *Handbook of Engineering Mechanics*, pg. 33-6, McGraw- Hill, 1962.
26. Timoshenko, S., and Gere, J., *Theory of Elastic Stability*, 2nd Edition, McGraw-Hill Book Company, New York, 1961.
27. Sauv , R.G., Teper, W., *Parametric Studies of Process Equipment Tube/Support Plate Impact Models*, Transactions of the 8th International Conference on Structural Mechanics in Reactor Technology, Brussels, August 1985.
28. Rogers, R.J., et al, *Fluid Effects on Multi-Span Heat Exchanger Tube Vibration*, Topics in Fluid Structure Interaction, ASME PVP Conference, San Antonio, 1984.
29. Shin, Y.S., et al, *Vibro Impact Responses of a Tube with Tube-Baffle Interaction*, ASME Paper 78-PVP-20, 1978.
30. Ma and Bathe, *On Finite Element Analysis of Pipe Whip Problems*, Nuclear Engineering and Design, Volume 37, 1976.

31. Cook, R., *Concepts and Applications of Finite Element Analysis*, 2nd Edition, John Wiley & Sons, Toronto, 1981.
32. Sauvé, R.G., *H3DMAP: Three Dimensional Matrix Analysis Program for Structures*, D&D Report No. 83509, Rev 1, August 1985.
33. Kanok-Nukulchai, W., *A Simple and Efficient Finite Element for General Shell Analysis*, International Journal for Numerical Methods in Engineering, Vol 14, pps. 179-200, 1979.
34. Kanok-Nukulchai, W., Taylor, R., Hughes, T., *A Large Deformation Formulation for Shell Analysis by the Finite Element Method*, Computers and Structures, Vol. 13, pp. 19-27, 1981.
35. Hughes, T., Liu, W., *Non-Linear Finite Element Analysis of Shells: Part I. Three Dimensional Shells*, Computer Methods in Applied Mechanics and Engineering, 26, 1981.
36. Bathe, K.J., Ramm, E., Wilson, E.L., *Finite Element Formulations for Large Deformation Dynamic Analysis*, International Journal for Numerical Methods in Engineering, Vol 9, 1975.
37. Karlsson and Hibbitt, *Analysis of Pipe Whip*, EPRI Report No. NP-1208, 1979.
38. Wouters, C., *Geometric Non-Linear 3-D Dynamic Analysis of Shells*, Computers and Structures, Vol 15, No. 6, 1982.
39. Phillips and Taylor, *Theory and Applications of Numerical Analysis*, Academic Press (London), 1973.
40. Timoshenko, S., and Woinowsky-Krieger, S., *Theory of Plates and Shells*, 2nd Edition, McGraw-Hill Book Company, New York, 1959.
41. Seireg, A., *Mechanical Systems Analysis*, International Textbook Company, Pennsylvania, 1969.
42. Belytschko, T., Tsay, C., *A Stabilization Procedure for the Quadrilateral Plate Element with One-Point Quadrature*, International Journal for Numerical Methods in Engineering, Vol 19, 405-419, 1983.
43. Sauvé, R.G., Teper, W., *Impact Simulation of Process Equipment Tubes and Support Plates - A Numerical Algorithm*, ASME Journal of Pressure Vessel Technology, Vol 109, Feb. 1987.
44. Hughes, T.J.R., Taylor, R.L., Kanoknukulchai, W., *A Simple and Efficient Finite Element for Plate Bending*, International Journal of Numerical Methods in Engineering, 11, 1977.
45. Sauvé, R.G., *H3DMAPT, Three-Dimensional Thermal Analysis Program For Shells*, Ontario Hydro D&D Report No. 87418, Oct. 1987.
46. Sauvé, R.G., *H3DMAP Version 2, Three-Dimensional Matrix Analysis Program for Structures Volumes 1 and 2*, Ontario Hydro D&D Report No. 86035, Feb. 1986.

47. Sauv , R.G., *H3DMAP Version 3.1, Three-Dimensional Matrix Analysis Program for Structures, Volumes 1 and 2*, Ontario Hydro D&D Report No. 87419, October 1987.
48. Roark, R.J., *Formulas for Stress and Strain*, 5th Edition, McGraw-Hill Book Co., New York, 1975.
49. Timoshenko, S., Young, D.H., and Weaver, W. Jr., *Vibration Problems in Engineering*, Fourth Edition, John Wiley & Sons, New York, 1974.
50. Witmer, E.A., Balmer, H.A., Leech, J.W., and Pian, T.H.H., *Large Dynamic Deformations of Beams, Rings, Plates, and Shells*, AIAA Journal, Vol. 1, No. 8, August 1963.
51. Moreno, L., Leech, J.W., and Witmer, E.A., *An Improved Numerical Calculation Technique for Large Elastic-Plastic Transient Deformations of Thin Shells, Part 2 - Evaluation and Applications*, Journal of Applied Mechanics, June 1971, pp. 429-436.
52. Belytschko et al., *Explicit Algorithms for the Non-Linear Dynamics of Shells*, Computer Methods in Applied Mechanics and Engineering, Vol. 42, 1984.
53. Duffey and Key, *Experimental-Theoretical Correlations of Impulsively Loaded Clamped Circular Plates*, Experimental Mechanics, June 1969, pp. 241-249.
54. Russel, J.H., *Tritiated D,O Transport Container Computer-Model Verification Punch Test No. 2*, OHRD Report No. 85-83-K, April 1985.
55. Sauv , R.G., and Morandin, G.D., *Tritiated Heavy Water Transportation Package: Analytical Assessment of Fluid-Structure Interaction Effects During Impact*, OHRD Report No. 88-208-K, November 1988.
56. Teper, W.W., and Sauv , R.G., *Simplified Method for Predicting Impact Loads of Solid-Walled Transportation Packagings for Radioactive Materials*, Journal of Pressure Vessel Technology, Vol. 111, August 11, 1989.
57. Sauv , R.G., *Efficient Shell Elements for Three Dimensional Non-Linear Explicit Transient Dynamic Analysis*, OHRD Report No. 89-222- K, January 1990.
58. Peterson, R.E., *Stress Concentration Design Factors*, John Wiley and Sons, 1953.
59. Sauv , R.G., Morandin, G.D., Nadeau, E., *Impact Simulation of Liquid-Filled Containers Including Fluid-Structure Interaction - Parts 1 and 2*, ASME PVP-Vol. 198, Advances in Dynamics of Piping and Structured Components, ASME PVP Nashville Conference, June, 1990.
60. Sauv , R.G., *A Contact-Impact Algorithm for Three-Dimensional Finite Deformation*, OHRD Report No. 90-314-K, January 1991.
61. Sauv , R.G., Dubey, R.N., *Efficient Shell Elements for Three-Dimensional Non-Linear Structural Dynamic Analysis*, Proceedings of the 4th International Conference on Recent Advances in Structural Dynamics, University of Southampton, U.K., 1991.

62. Misra, A., Nguyen, V., *Impact of a Long Radius Piping Elbow on a Rigid Wall*, ASME, 1991.
63. Sauvé, R.G., Metzger, D.R., Byrne, T.P., *Three Dimensional Analytical Modelling of Rolled Joint Residual Stress - Feasibility Study*, OHRD Report No. 90-315-P (COG-90-319), December 1990.
64. Shah, N.N., Zane, R.M., *Results of 1/4 Scale Radioactive Filter Transportation Package Model 9m Drop Tests*, OHRD Report No. 87-182-K, January 1988.
65. Dubey, R.N., Sauvé, R.G., *Numerical Integration of Rate Constitutive Equations Under Finite Deformation*, Proceedings of the 11th Structural Mechanics in Reactor Technology Conference, Paper B02/4, Japan, 1991.

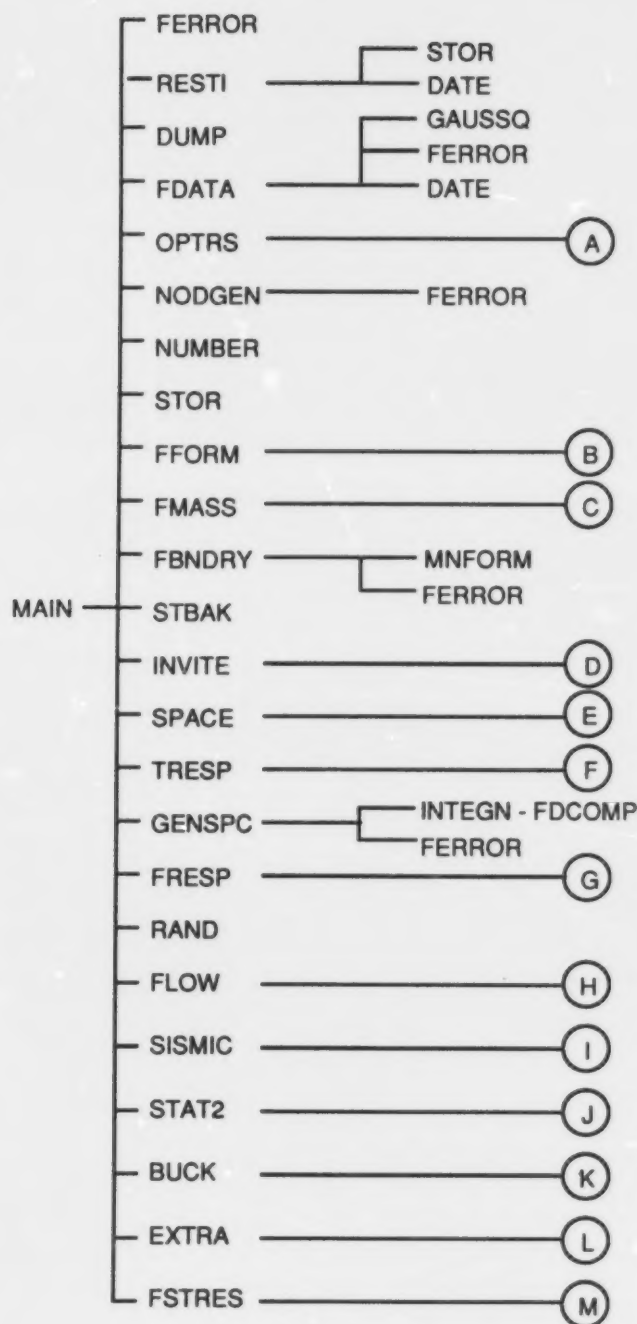
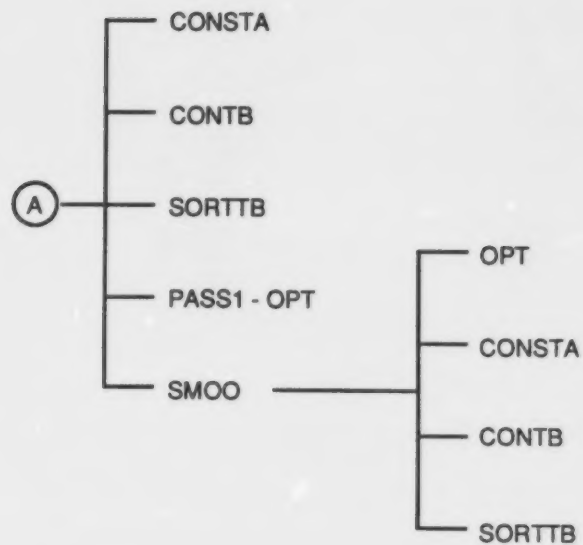


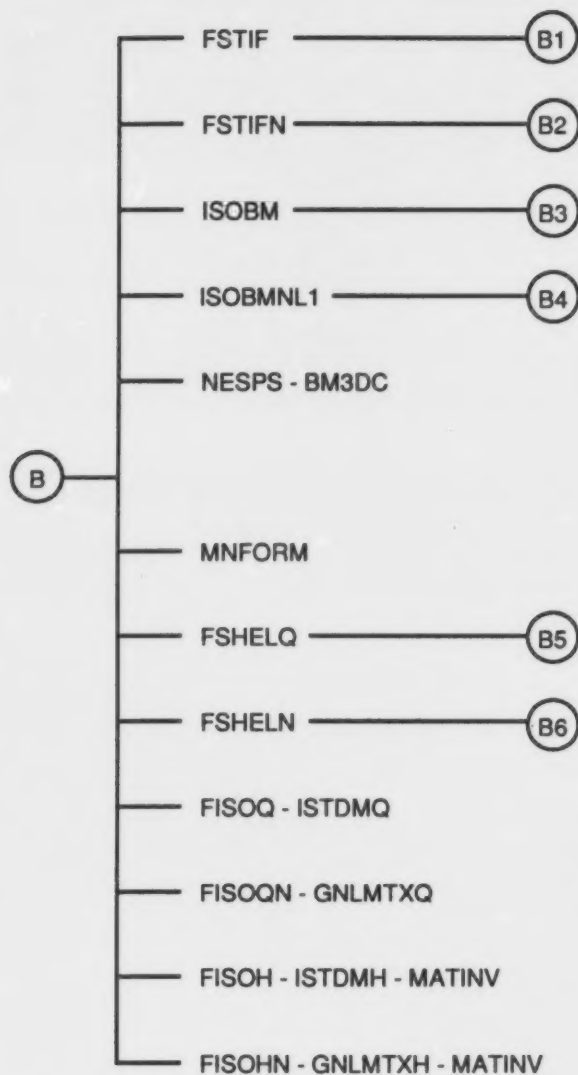
FIGURE 1.1

H3Dmap VRS. 4 OVERALL PROGRAM STRUCTURE MODULES



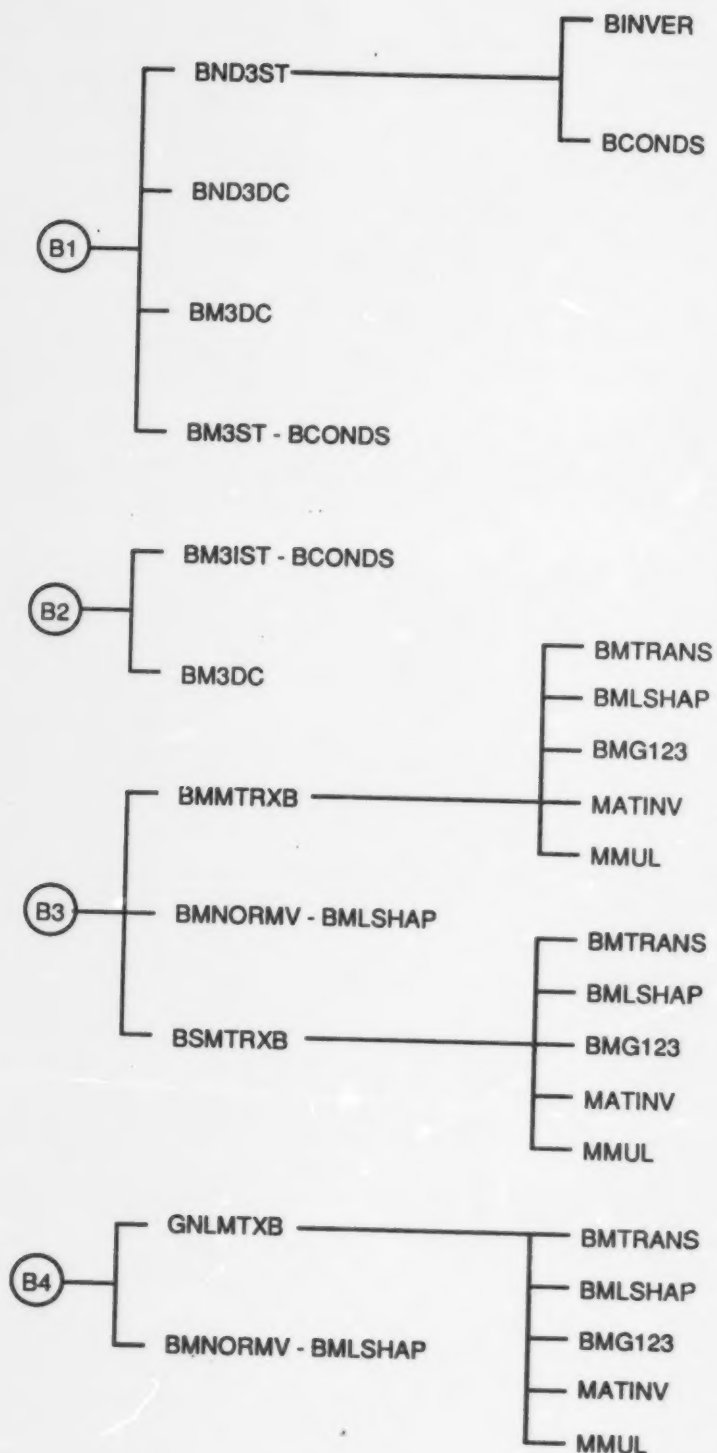
NODE NUMBERING OPTIMIZATION ROUTINES

FIGURE 1.2
H3DMAP STRUCTURE



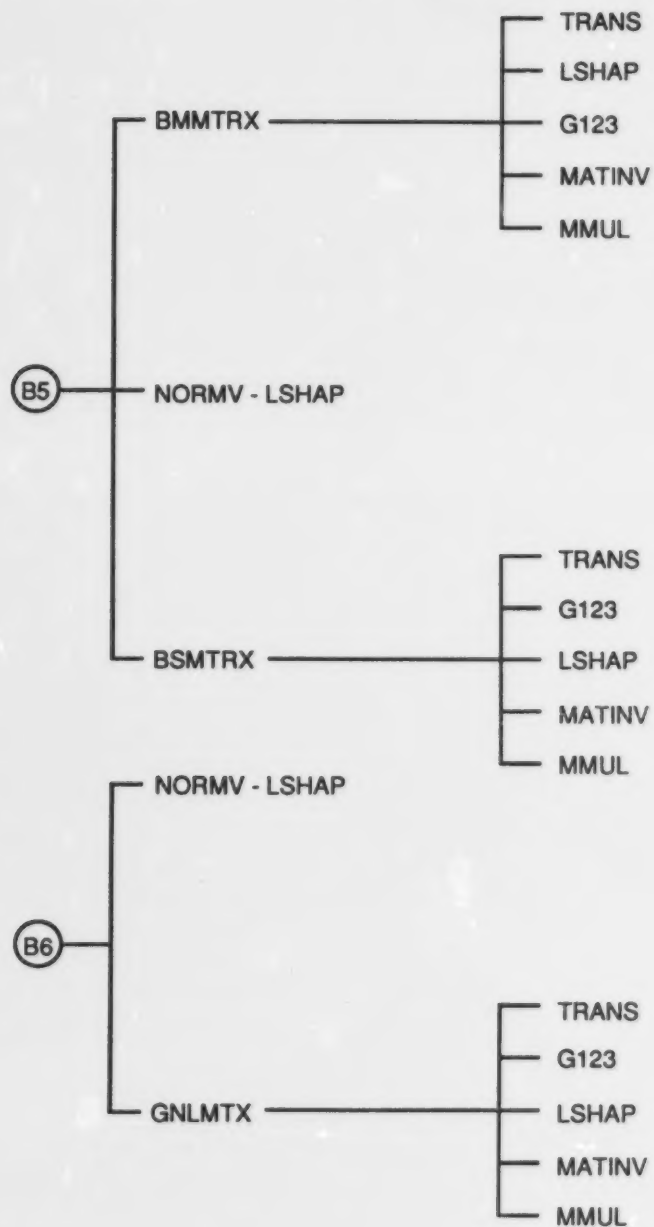
ASSEMBLY, ELEMENT ROUTINES

FIGURE 1.3
H3DMAP STRUCTURE



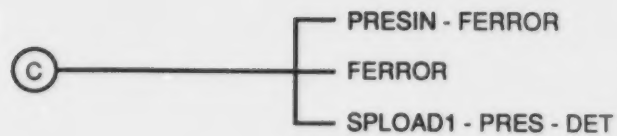
CUBIC AND ISOPARAMETRIC BEAM ELEMENT ROUTINES

FIGURE 1.4
H3DMAP STRUCTURE

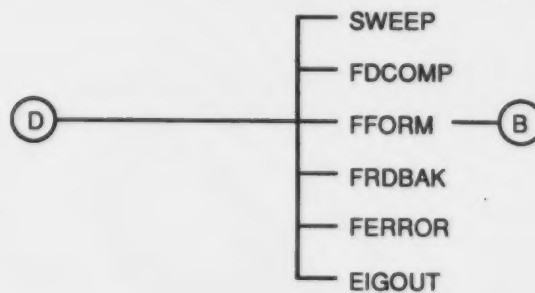


SHELL ELEMENT ROUTINES

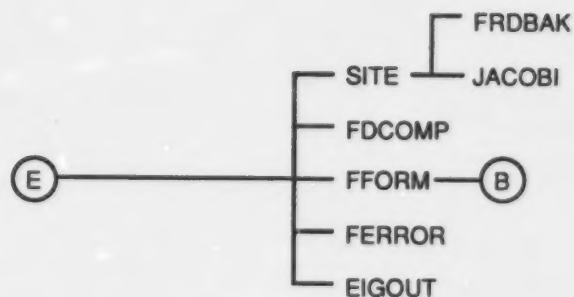
FIGURE 1.5
H3DMAP STRUCTURE



LOAD MASS ROUTINES

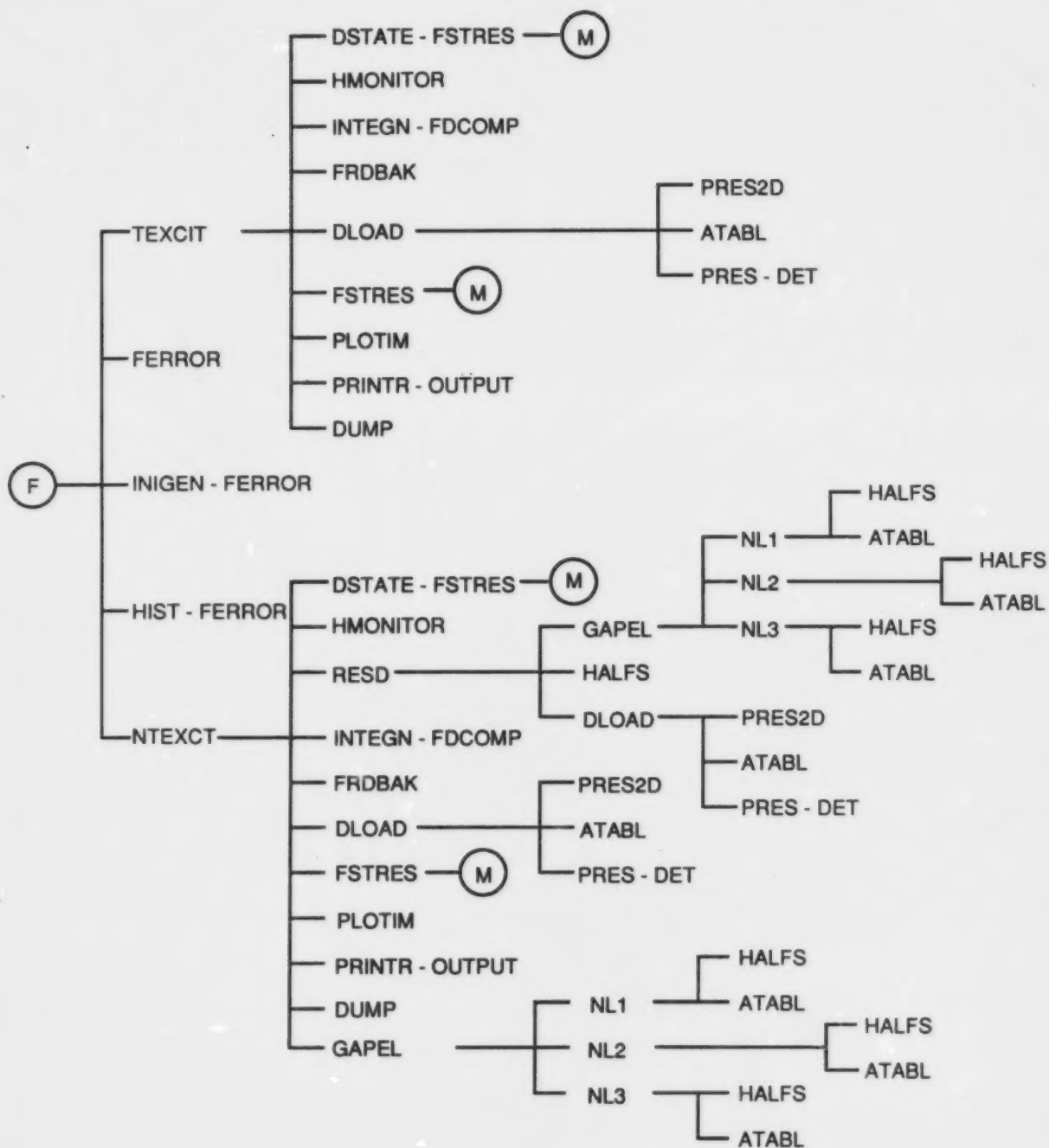


INVERSE ITERATION ROUTINES



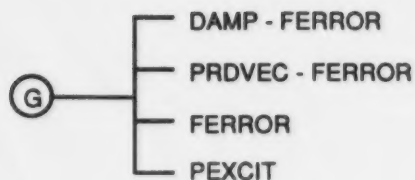
SUBSPACE ITERATION ROUTINES

FIGURE 1.6
H3DMAP STRUCTURE

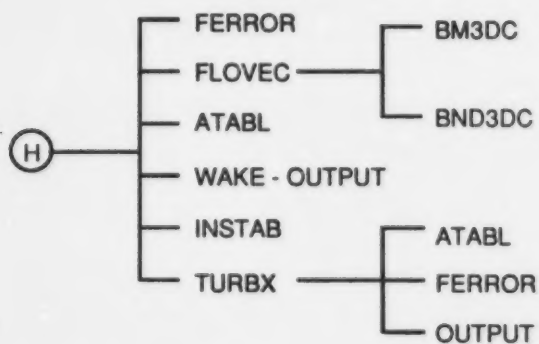


IMPLICIT DYNAMIC TRANSIENT ANALYSIS ROUTINES

FIGURE 1.7
H3DMAP STRUCTURE



FREQUENCY RESPONSE ANALYSIS ROUTINES

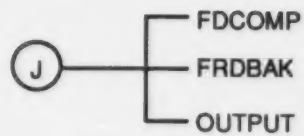


FLOW INDUCED VIBRATION ANALYSIS ROUTINES

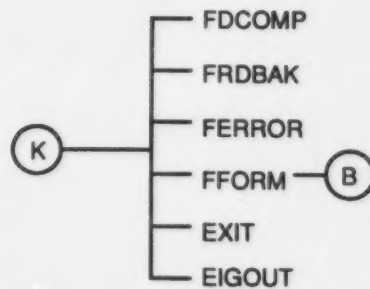


SEISMIC RESPONSE ANALYSIS ROUTINES (RESPONSE SPECTRUM METHOD)

FIGURE 1.8
H3DMAP STRUCTURE

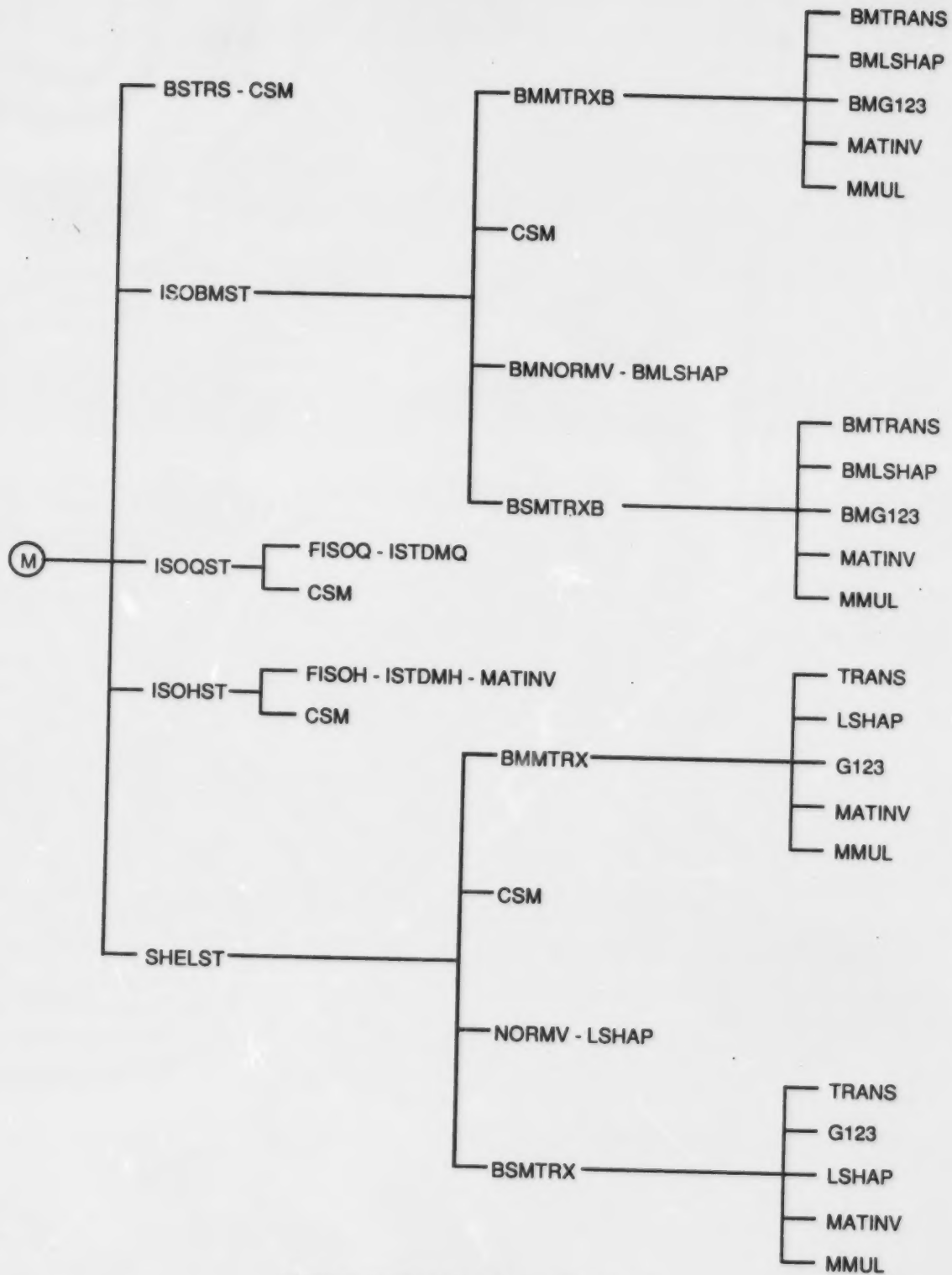


LINEAR STATIC ANALYSIS ROUTINES



STRUCTURAL STABILITY ROUTINES

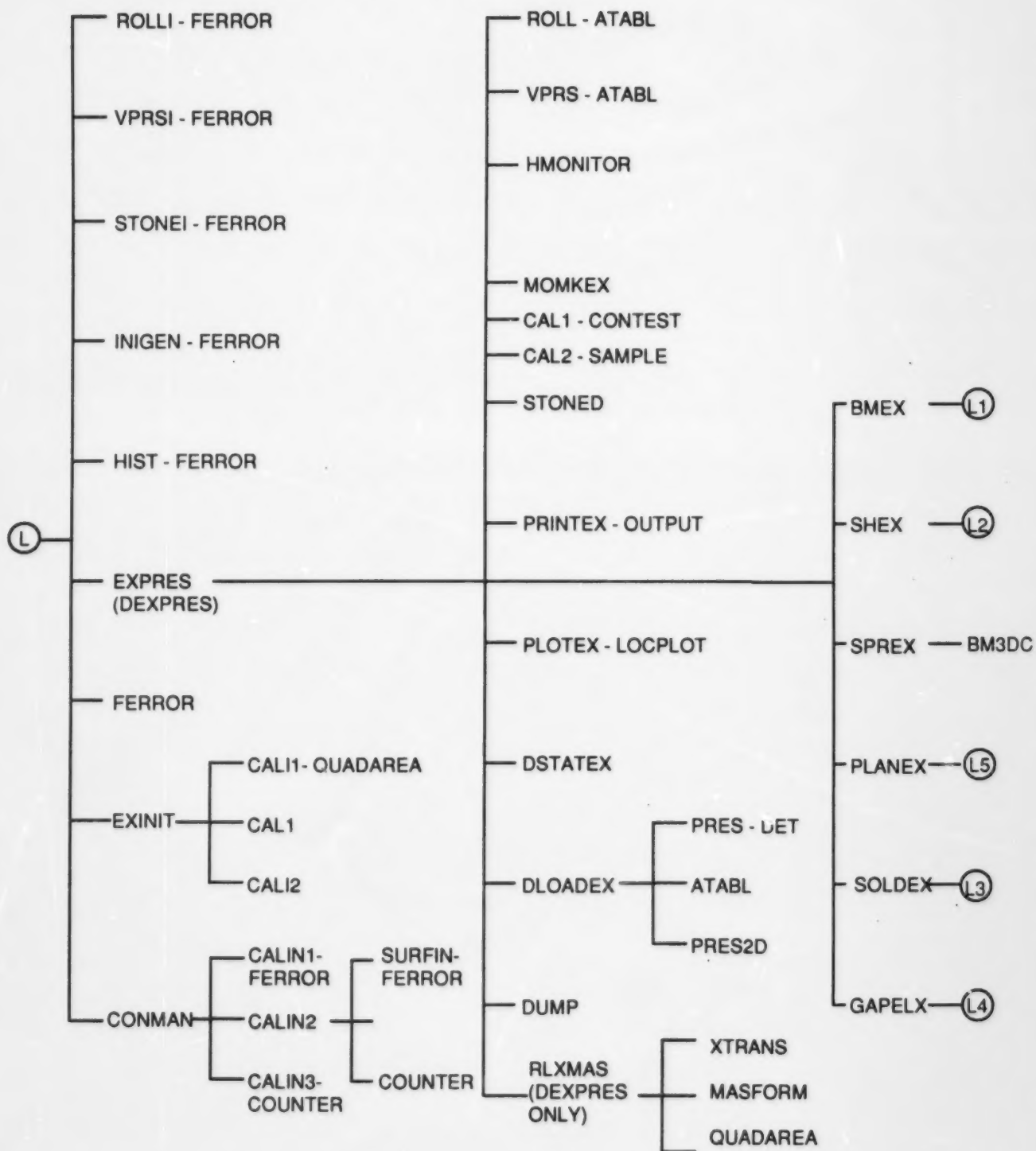
FIGURE 1.9
H3DMAP STRUCTURE



LINEAR STRESS RECOVERY ROUTINES

FIGURE 1.10

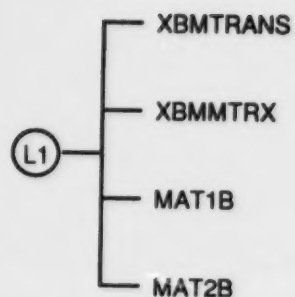
H3DMAP STRUCTURE



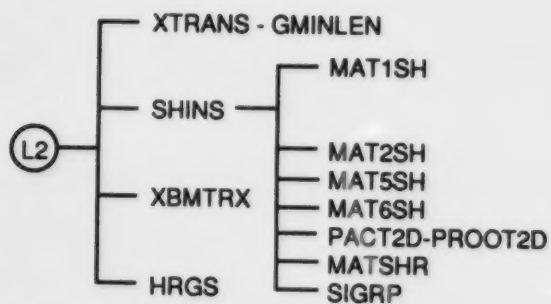
EXPLICIT TRANSIENT DYNAMIC ANALYSIS ROUTINES

FIGURE 1.11

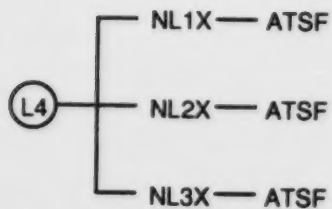
H3DMAP STRUCTURE



EXPLICIT BEAM ELEMENT ROUTINES

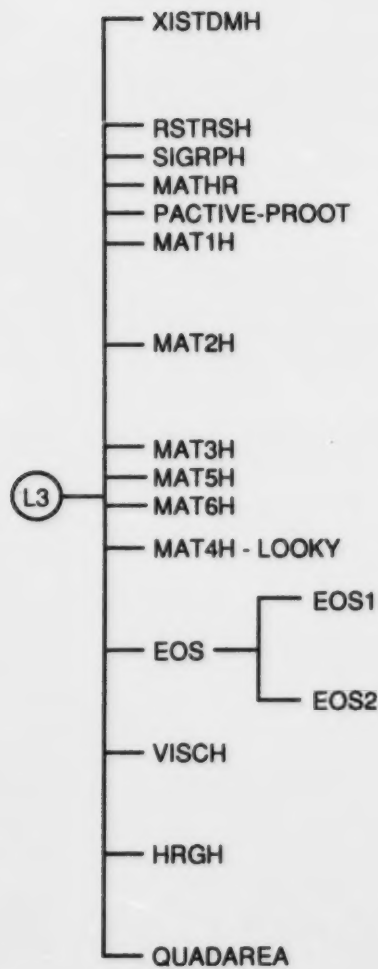


EXPLICIT SHELL ELEMENT ROUTINES



EXPLICIT ANALOG ELEMENT ROUTINES

FIGURE 1.12
H3DMAP STRUCTURE



EXPLICIT THREE DIMENSIONAL CONTINUUM ELEMENT ROUTINES

FIGURE 1.13
H3DMAP STRUCTURE

COMPUTER CODE H3DMAP

Non-linear Continuum/Structural Mechanics

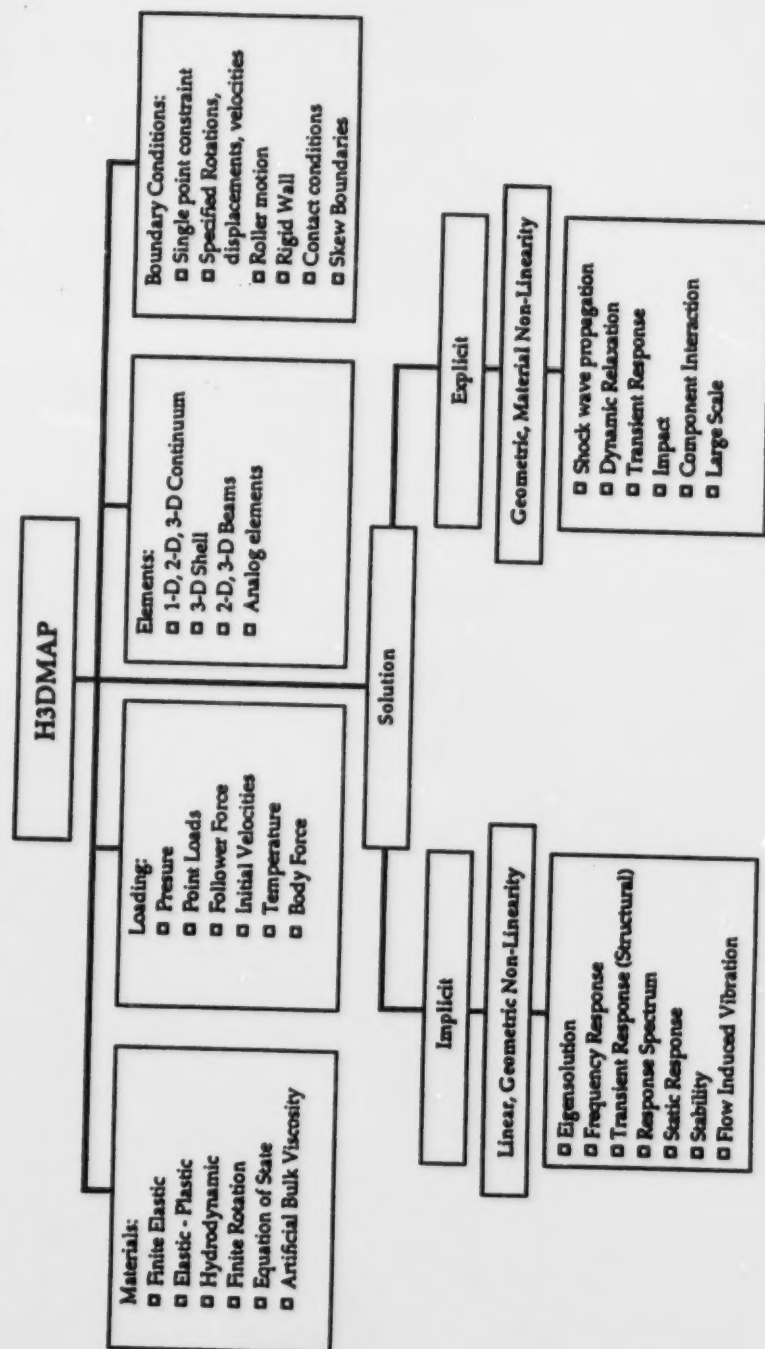
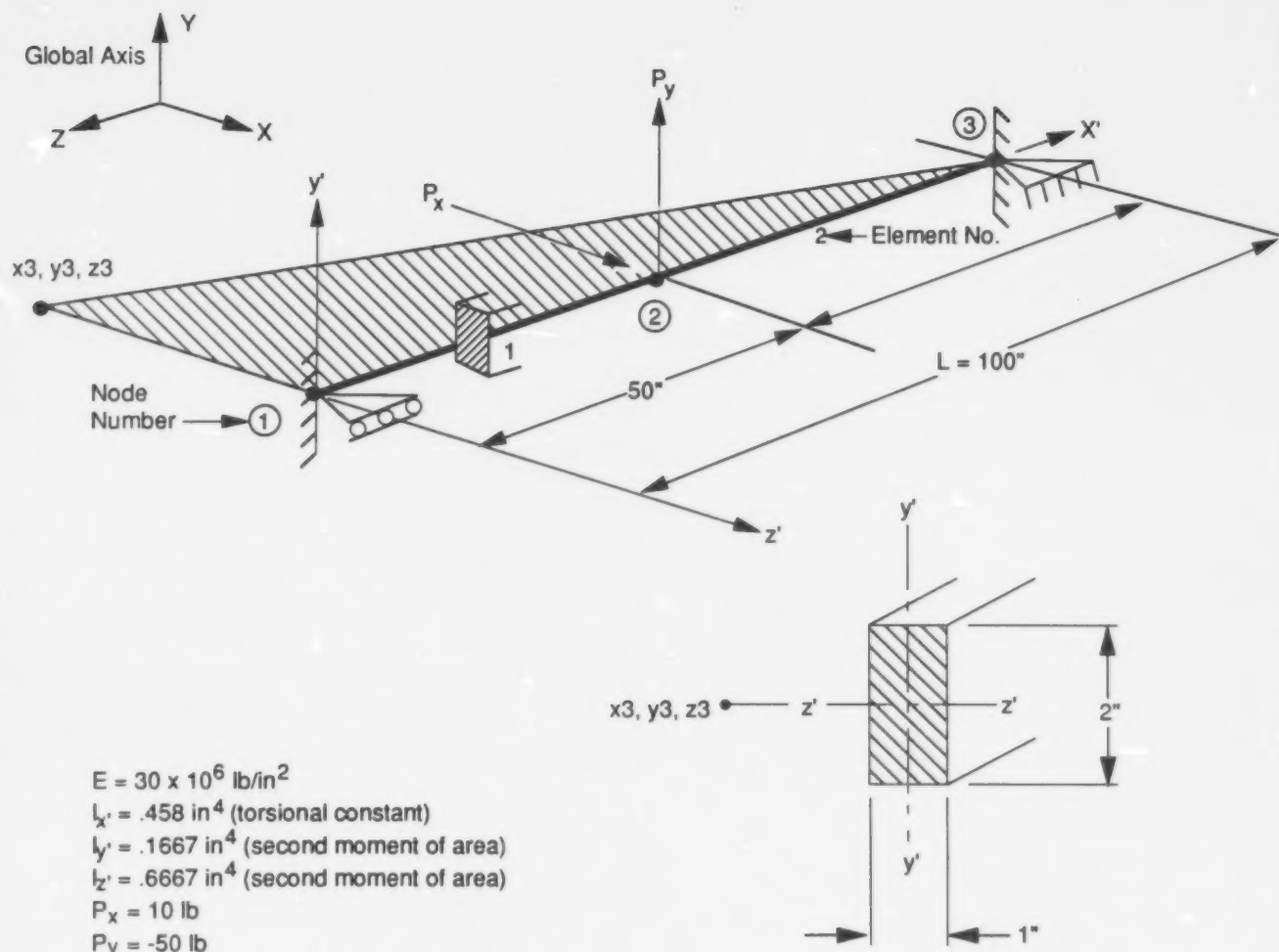


FIGURE 1.14

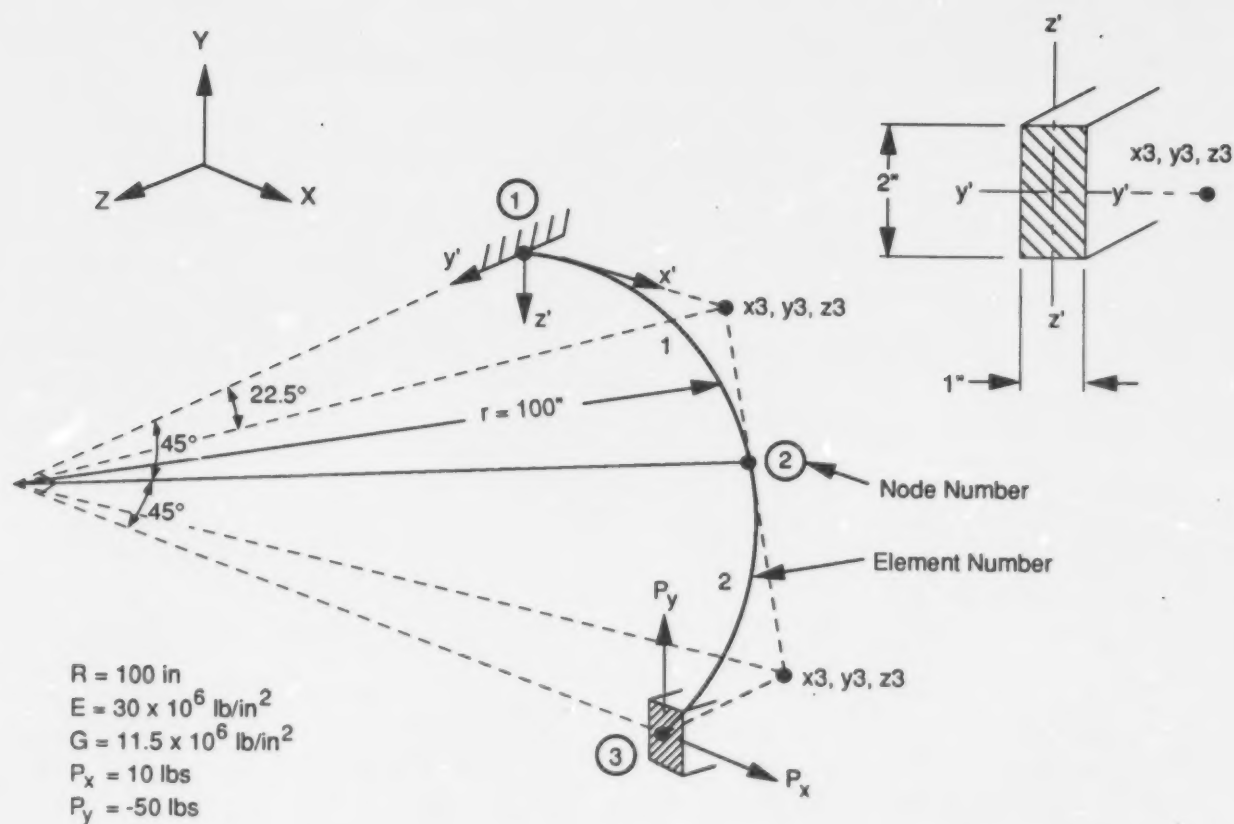
OVERALL COMPUTER CODE CAPABILITIES



Node No. 2	Analytical Results	Computer Model Results
Global Y-Displacement (in)	-.01302	-.01302
Max. Local Z-Axis Moment (lb-in) (Centre & Ends)	625	625
Global X-Displacement (in)	.04167	.04167
Max. Local Y-Axis Moment (lb-in) (Centre)	250	250

FIGURE 2.1.1

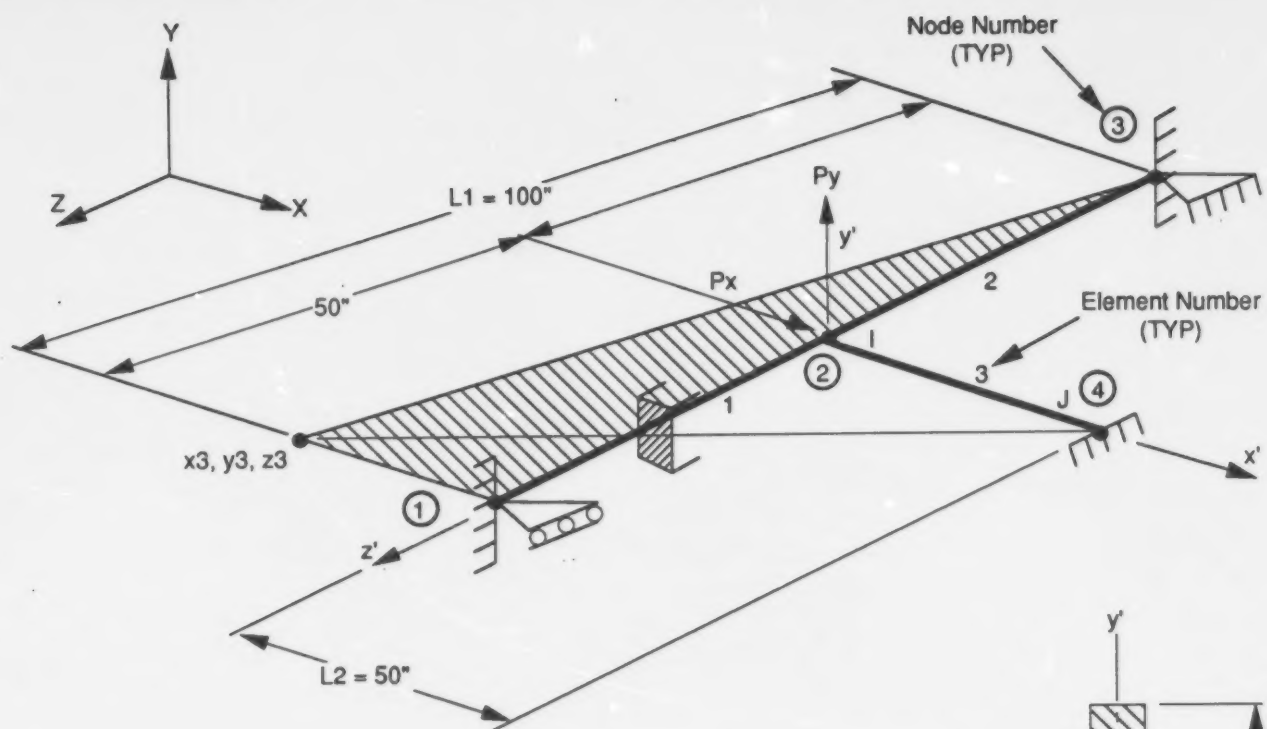
CASE 1: COMPUTER MODEL (STRAIGHT BEAM TEST)



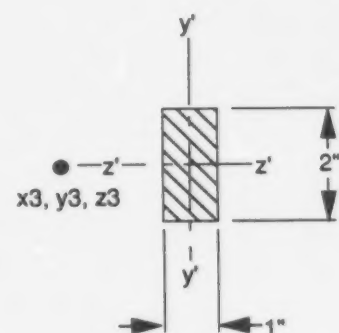
	Analytical Results	Computer Model Results
Global X-Direction Displacement (in) (Node #3)	1.57	1.57
Global Y-Direction Displacement (in) (Node #3)	-5.3448	-5.3335
Global Z-Direction Displacement (in) (Node #3)	-1.00	-1.00
Max. Local Torsional Moment (lb-in) (Element #1)	-5000	-5000
Max. Local Y-Axis Moment (lb-in) (Element #1)	5000	5000
Max. Local Z-Axis Moment (lb-in) (Element #1)	1000	1000

FIGURE 2.2.1

CASE 2: COMPUTER MODEL (CURVED BEAM TEST)



$E = 30 \times 10^6 \text{ lb/in}^2$
 $I_{zz} = .66667 \text{ in}^4$
 $I_{yy} = .166667 \text{ in}^4$
 $I_{xx} = .458 \text{ in}^4$
 $A = 2.0 \text{ in}^2$
 $P_y = -50 \text{ lbs}$
 $P_x = 10 \text{ lbs}$



	Analytical Results	Computer Model Results
Global X-Direction Displacement (in) (Node #2)	8.3317×10^{-6}	8.3317×10^{-6}
Global Y-Direction Displacement (in) (Node #2)	-.01157	-.01157
Local Z-Axis Moment (lb-in) (Node #4)	-277.8	-277.8

FIGURE 2.3.1

CASE 3: COMPUTER MODEL (MEMBER RELEASE TEST)

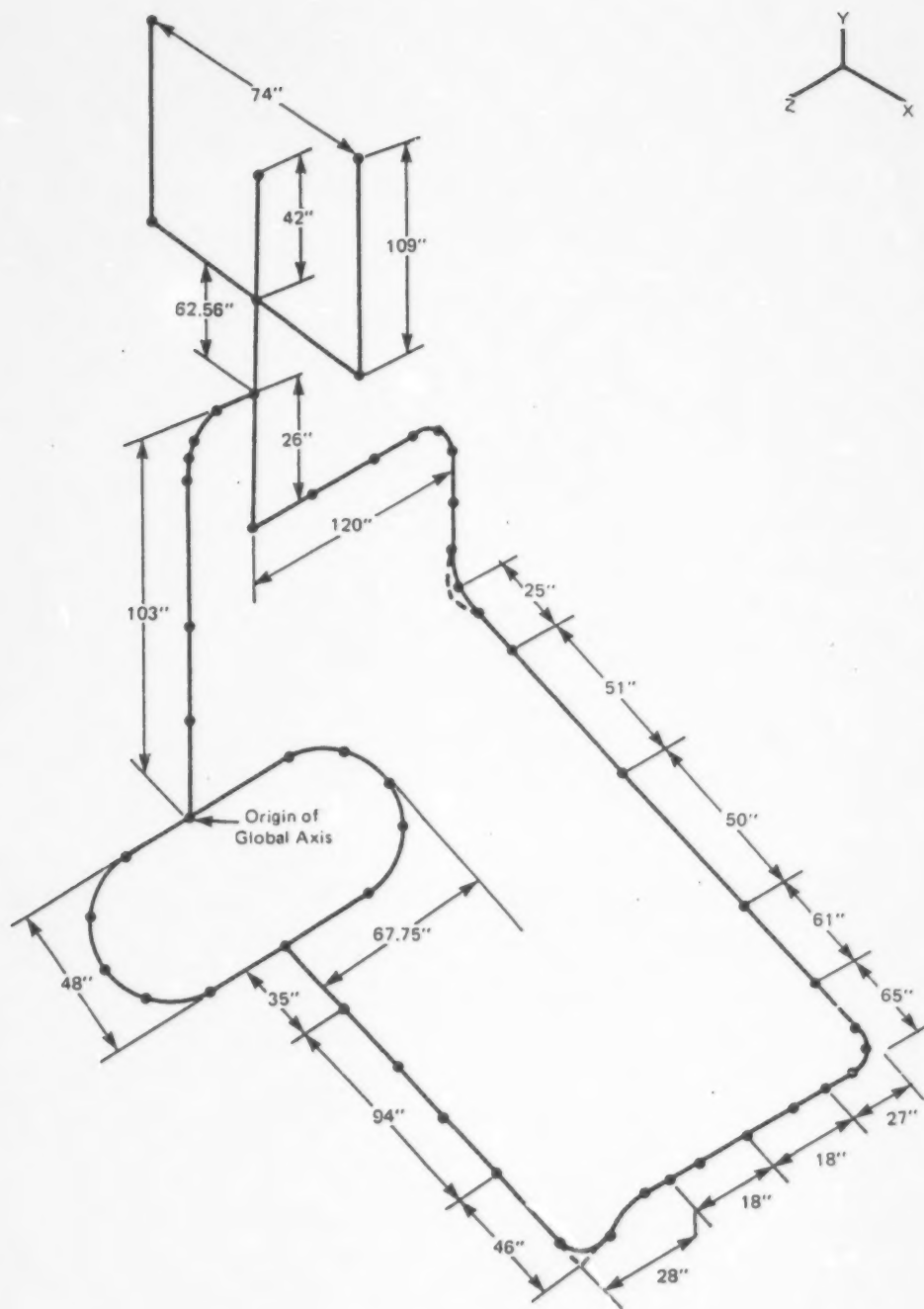


FIGURE 2.4.1
Case 4: Major Geometric Dimensions of Computer Model
for the Piping Test Loop

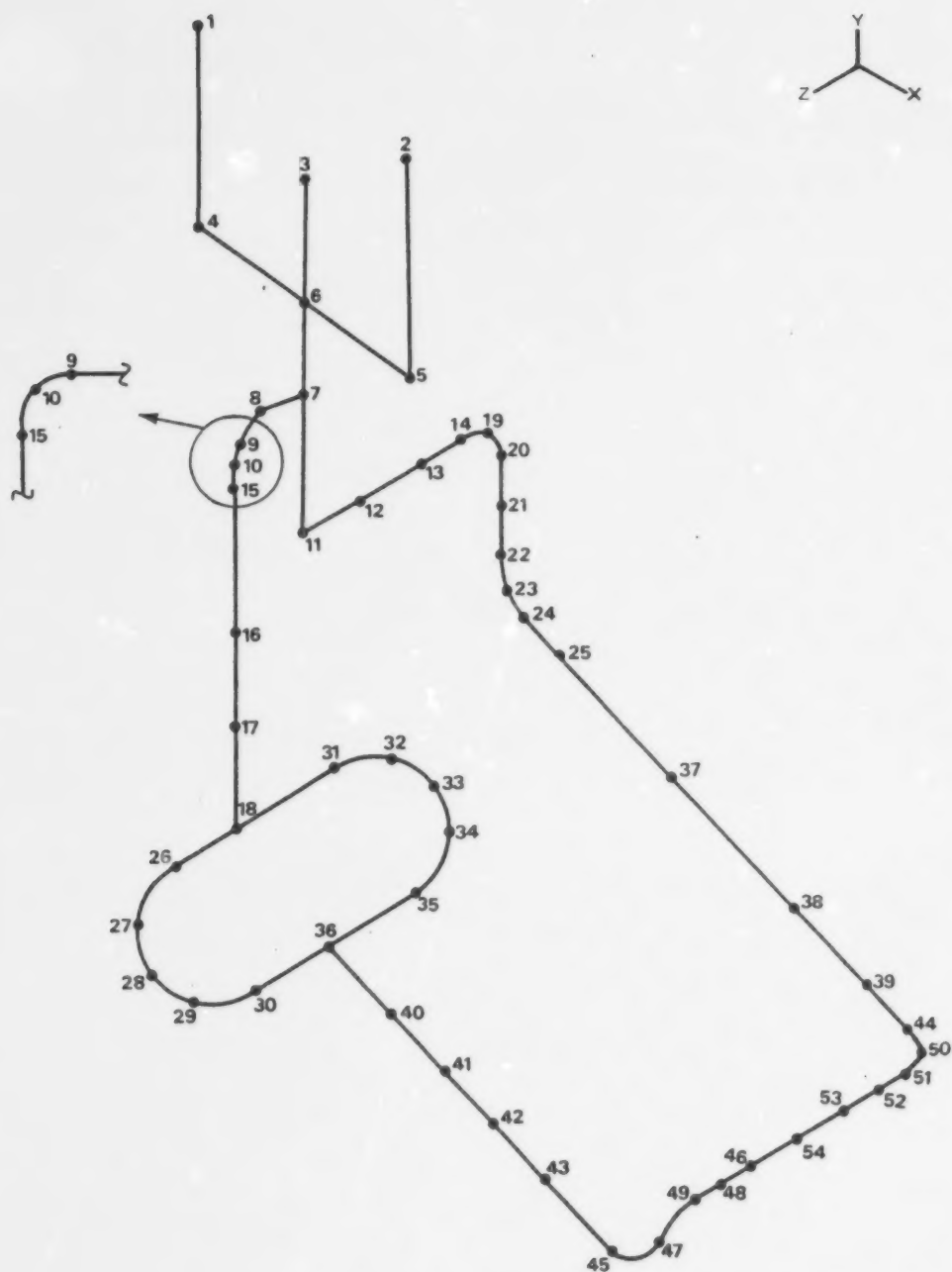


FIGURE 2.4.2
Case 4: Node Numbers of Computer Model
for the Piping Test Loop

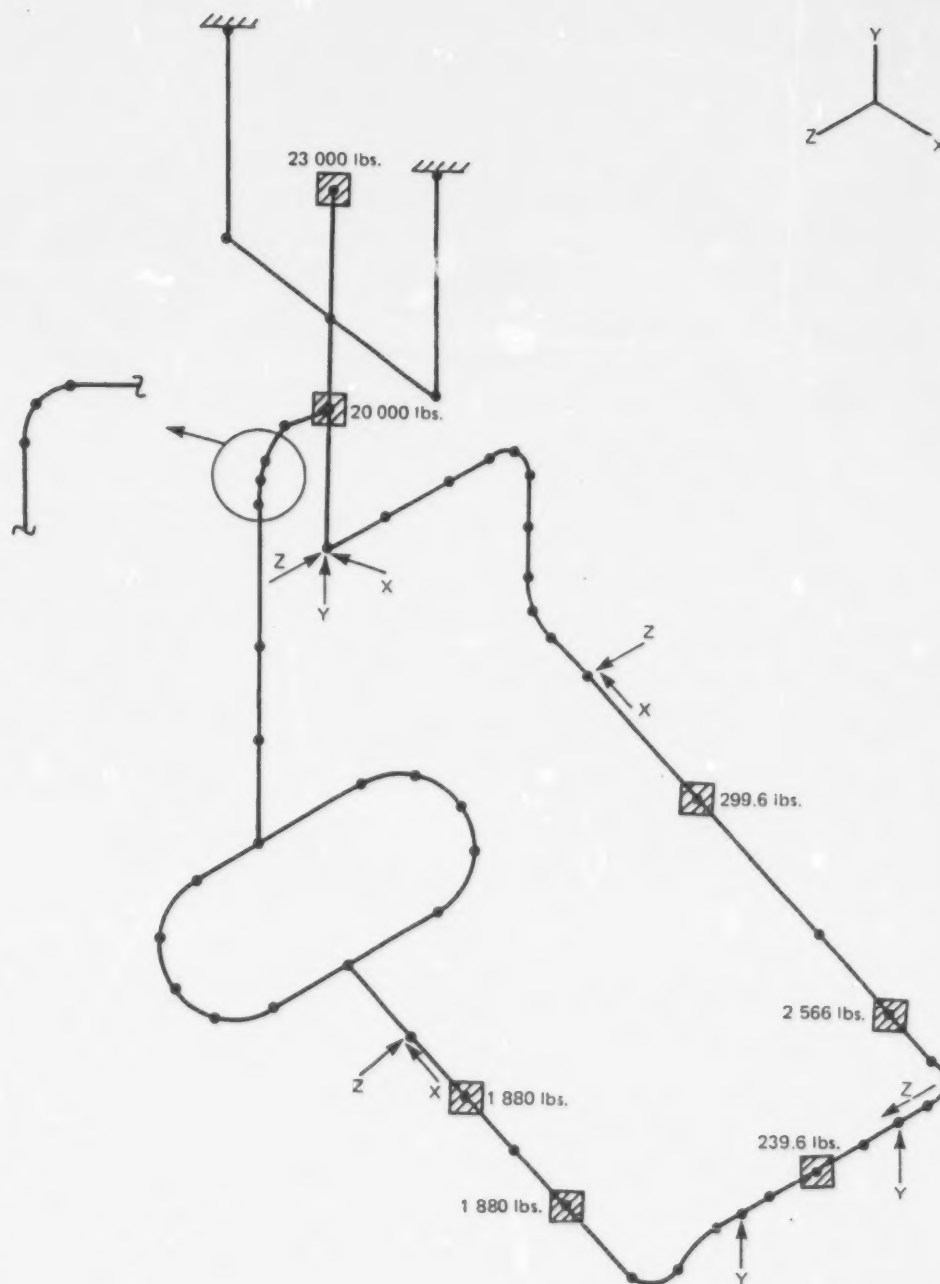


FIGURE 2.4.3
Case 4: Location of Concentrated Masses and Pipe
Restraint Applied to Computer Model
for the Piping Test Loop

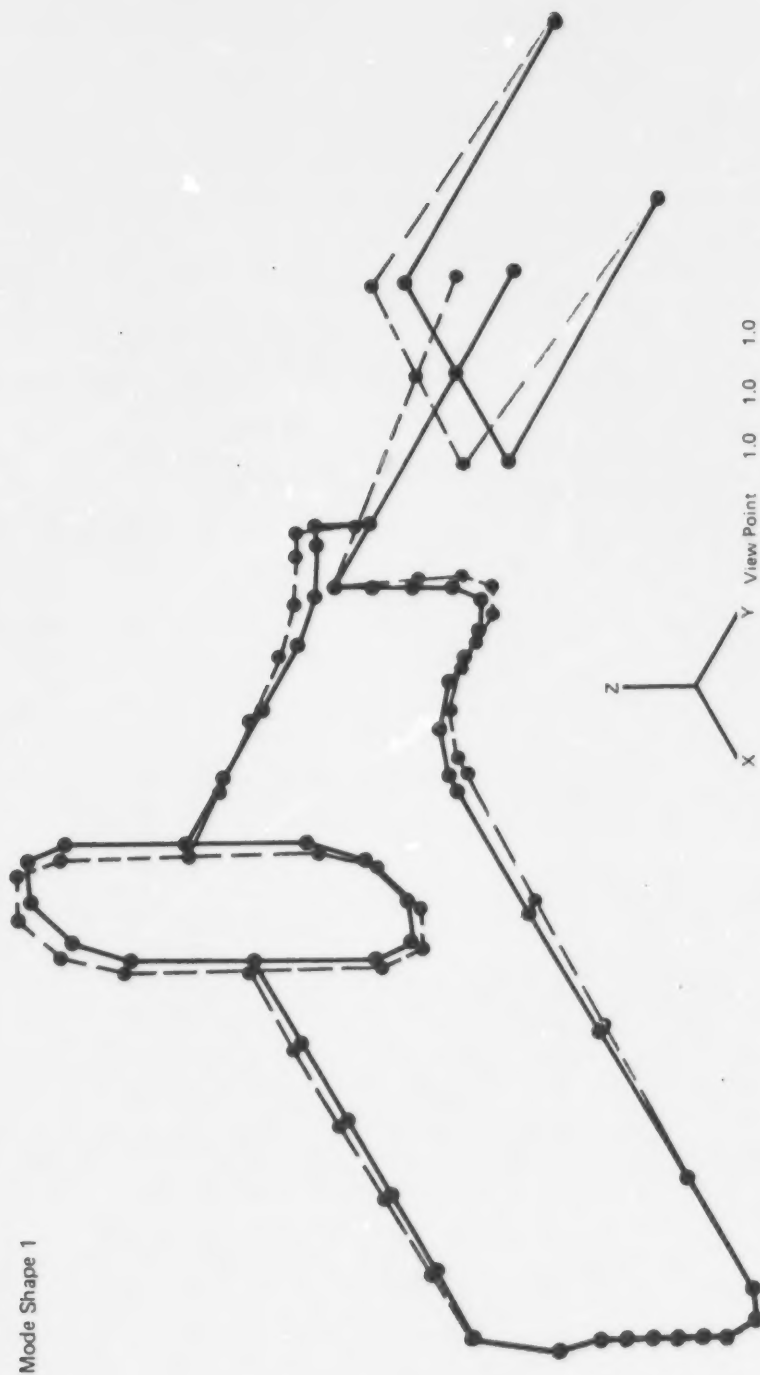


FIGURE 2.4.4
Case 4: Verification Test 2-PHT Pump Test Loop -- BDC Set 2

Mode Shape 2

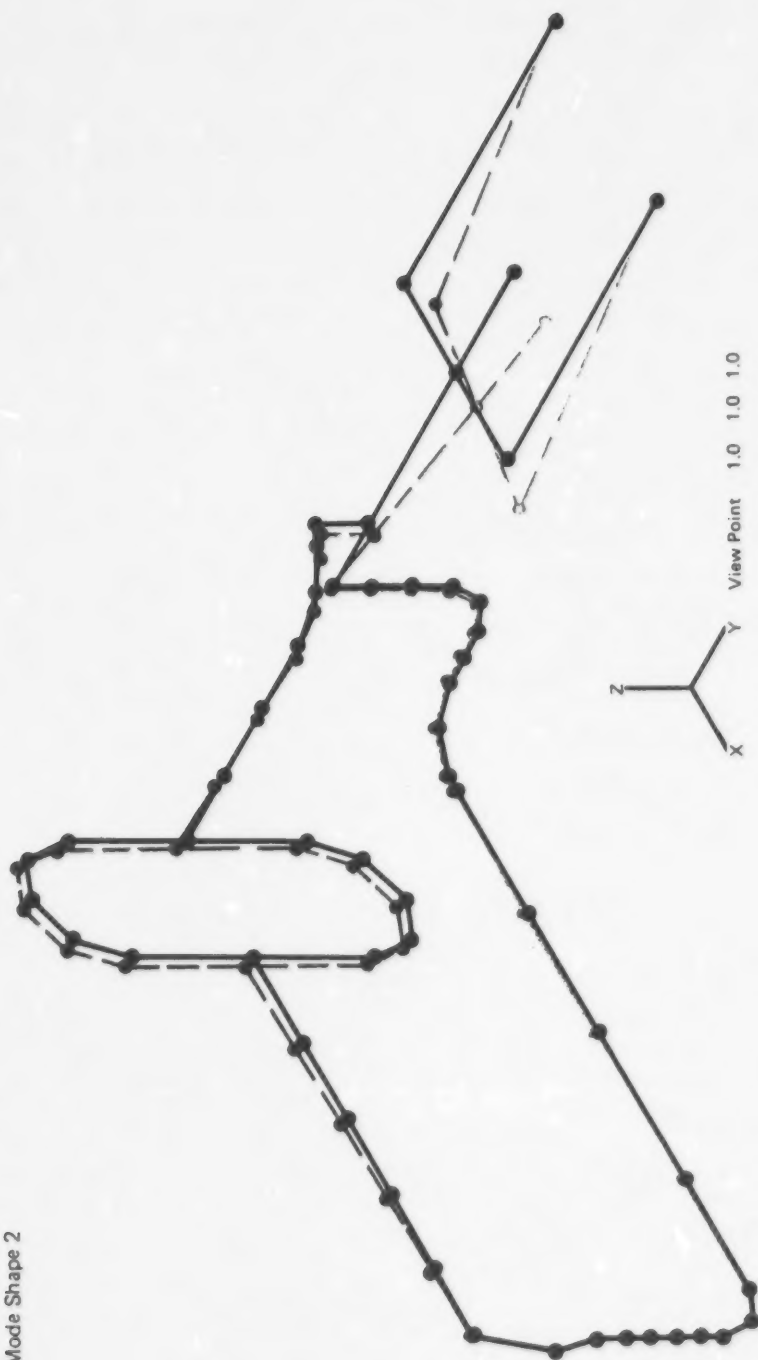


FIGURE 2.4.5
Case 4: Verification Test 2-PHT Pump Test Loop - BDC Set 2

Mode Shape 3

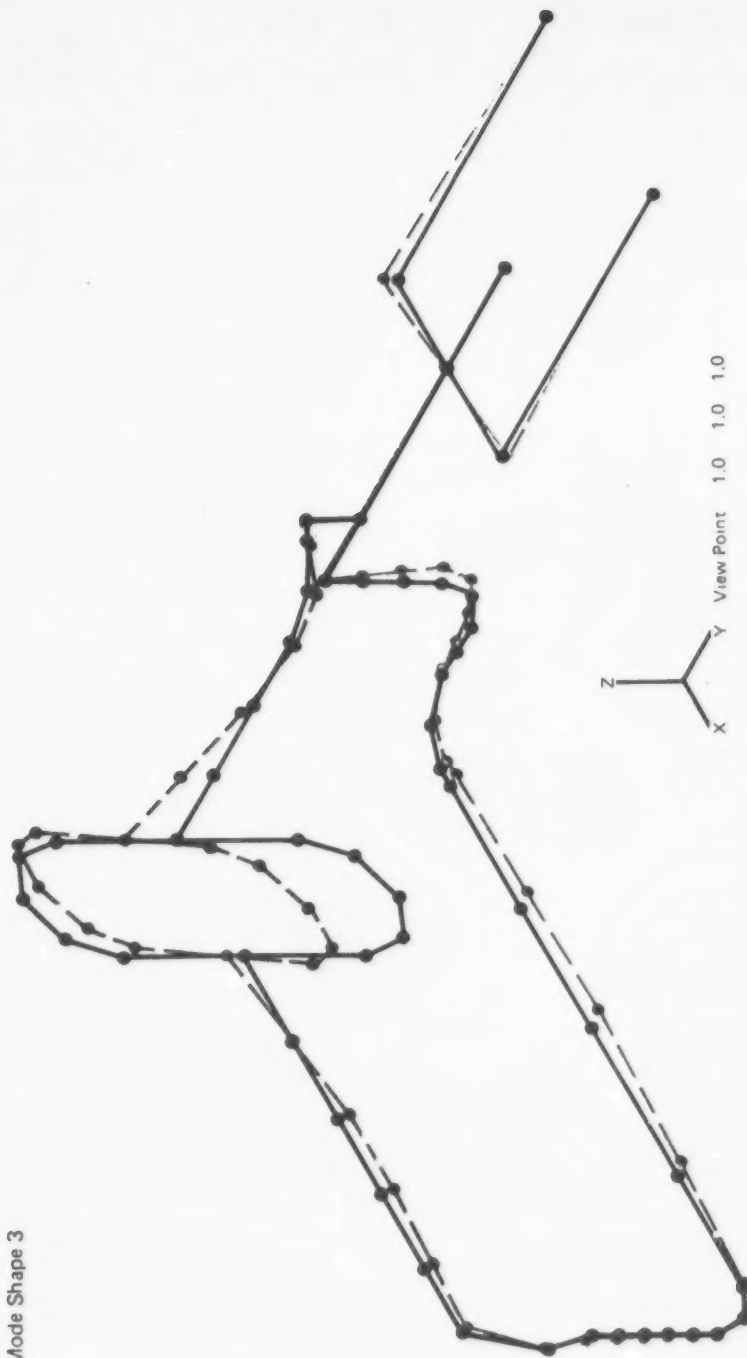


FIGURE 2.4.6
Case 4: Verification Test 2-PHT Pump Test Loop -- BDC Set 2

Mode Shape 5

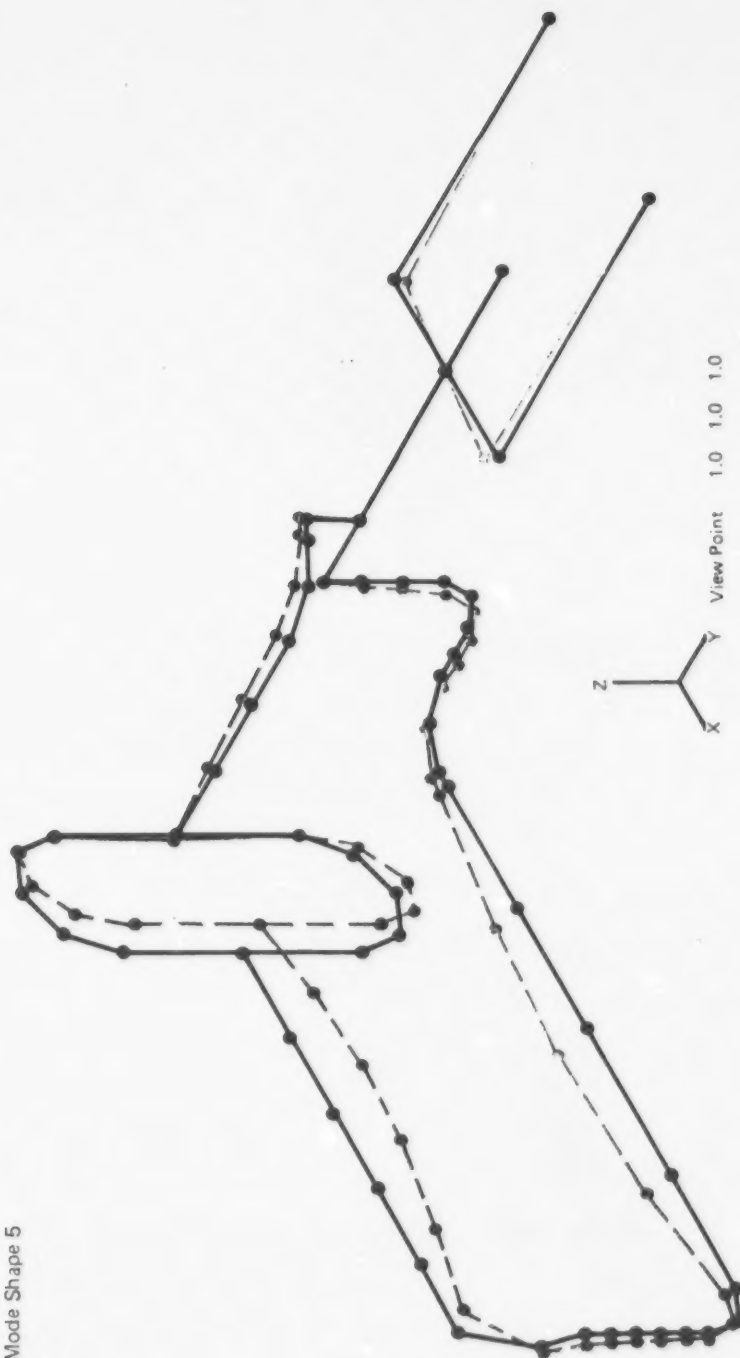
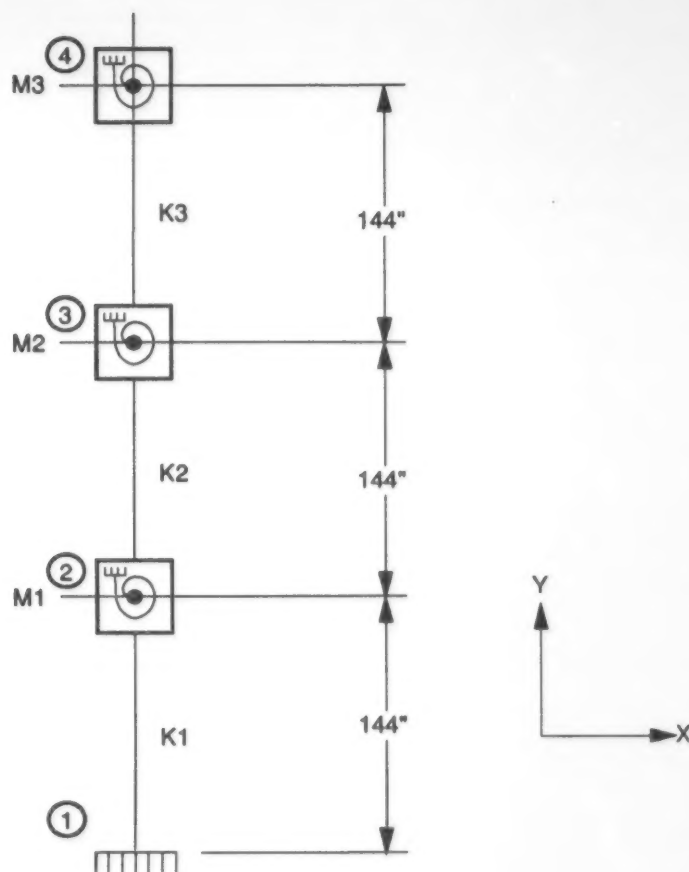


FIGURE 2.4.7
Case 4: Verification Test 2-PHT Pump Test Loop - BDC Set 2

$$\begin{aligned}
 M1 &= M2 = 8 \text{ Kip-sec}^2/\text{in}^2 \\
 &= 3091 \text{ Kips} \\
 M3 &= 4 \text{ Kip-sec}^2/\text{in}^2 \\
 &= 1545.6 \text{ Kips} \\
 E &= 30 \times 10^3 \text{ Kips/in}^2 \\
 K1 &= 1500 \text{ Kips/in} \\
 K2 &= 1000 \text{ Kips/in} \\
 K3 &= 500 \text{ Kips/in}
 \end{aligned}$$



	Analytical Results	Computer Model Results
Fundamental Frequency (HZ)	.996	.997
Second Natural Frequency (HZ)	2.14	2.18
Third Natural Frequency (HZ)	3.18	3.18

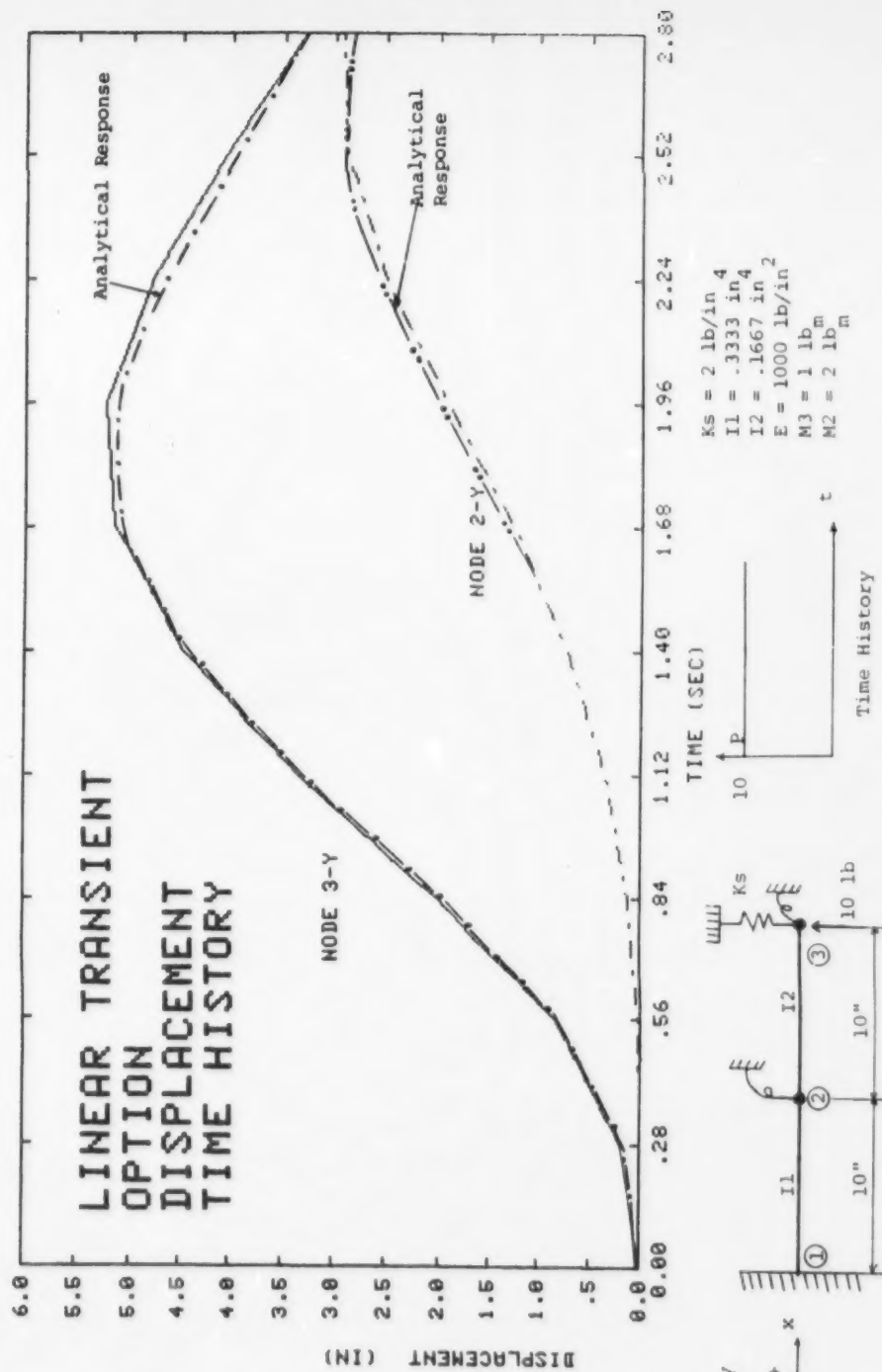
Node. No.	R.S.S. Accel (X-Dir)	R.S.S. Displ. (X-Trans)
1	.100	0.0
2	.390	.732
3	.336	1.100
4	.528	1.720

FIGURE 2.5.1

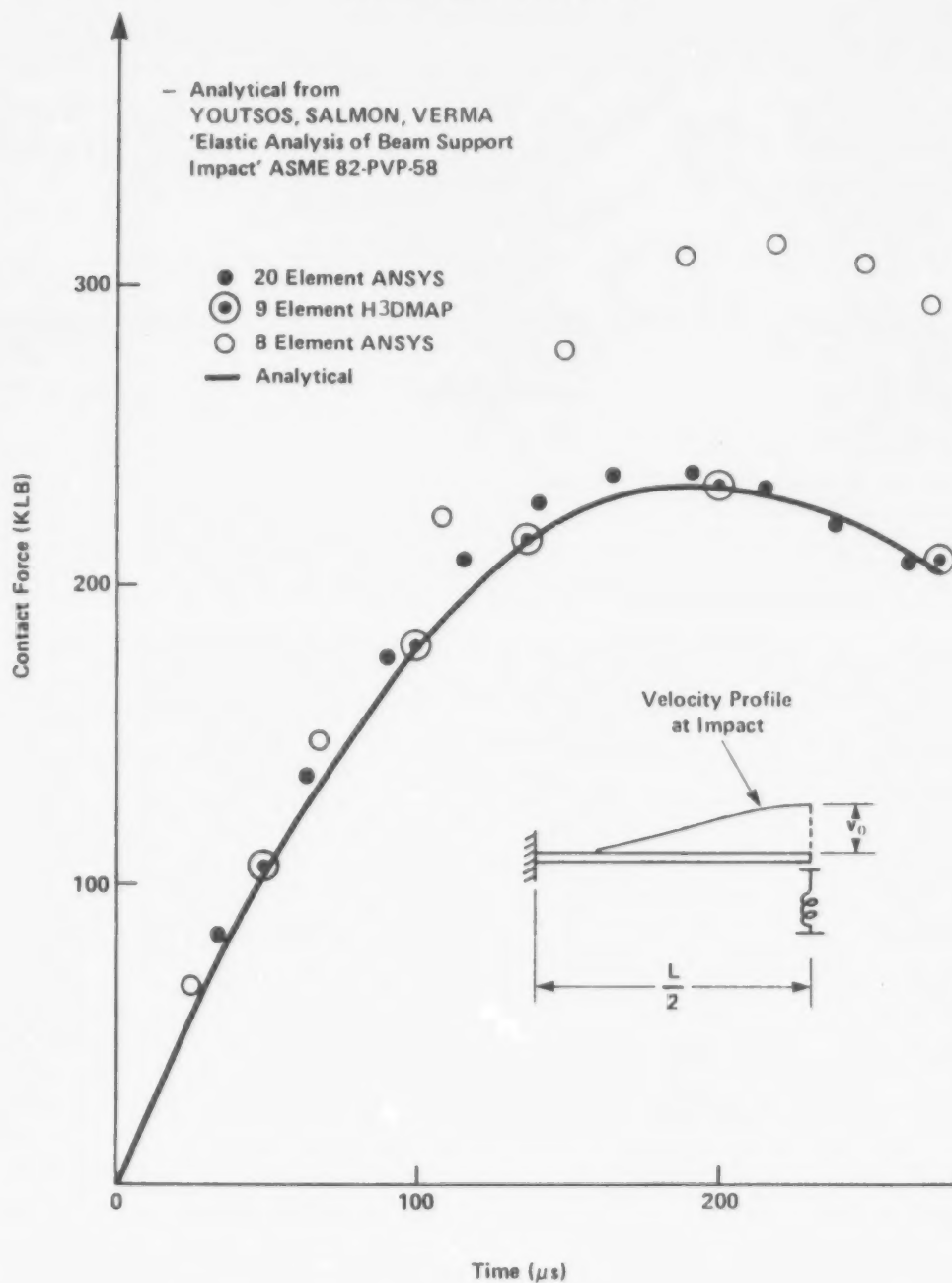
CASE 5: COMPUTER MODEL (EIGEN SOLUTION, SEISMIC TEST)

180984-RS-RD

89-221



CASE 8.1: H³D MAP COMPARISON WITH ANALYTICAL RESULTS
FOR INITIAL VELOCITY



$$NT = 21$$

$$K = .60 \times 10^8 \text{ lb/in}$$

FIGURE 2.8.1

CASE 8.2 : H3DMAP COMPARISON WITH ANALYTICAL RESULTS
FOR INITIAL VELOCITY

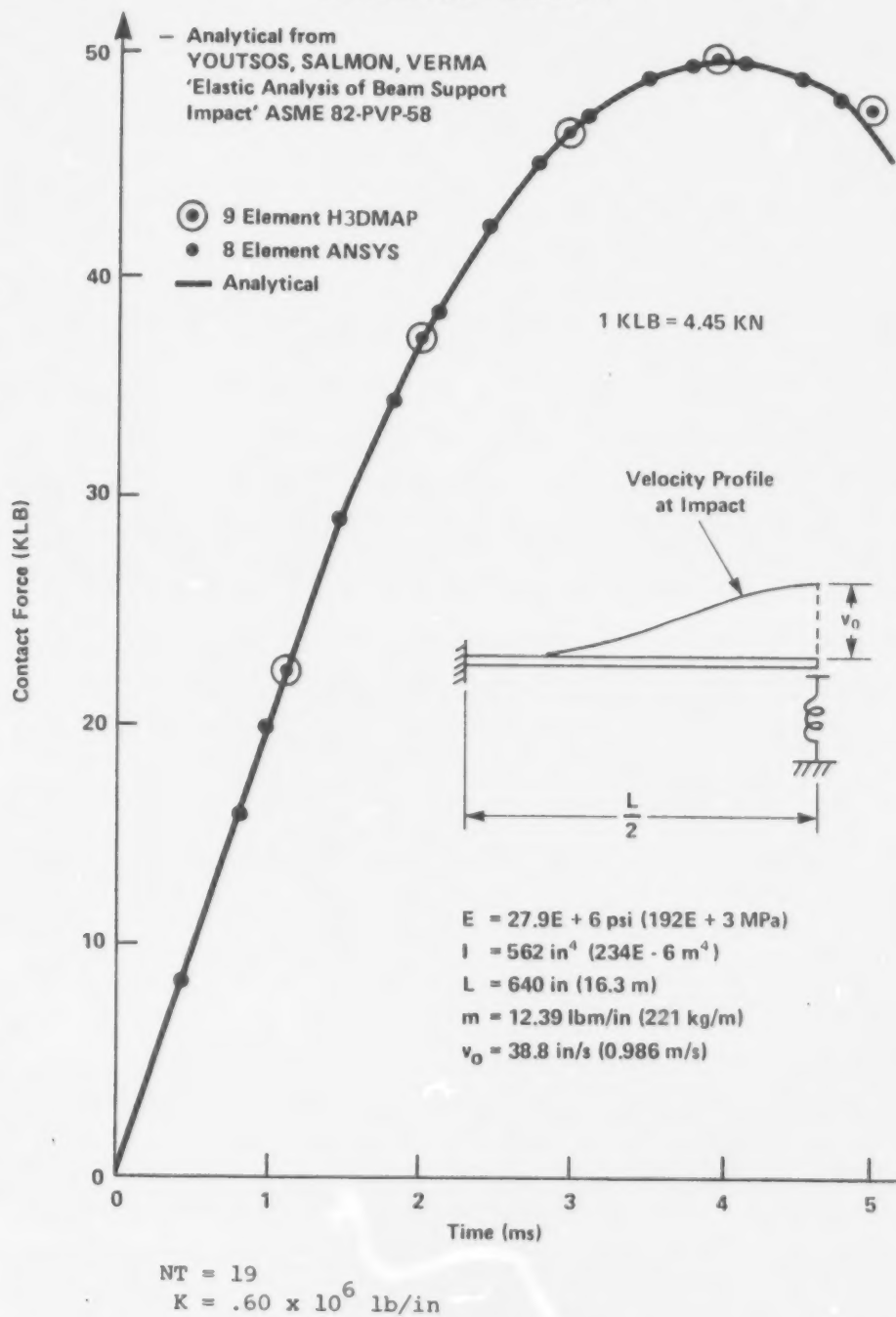
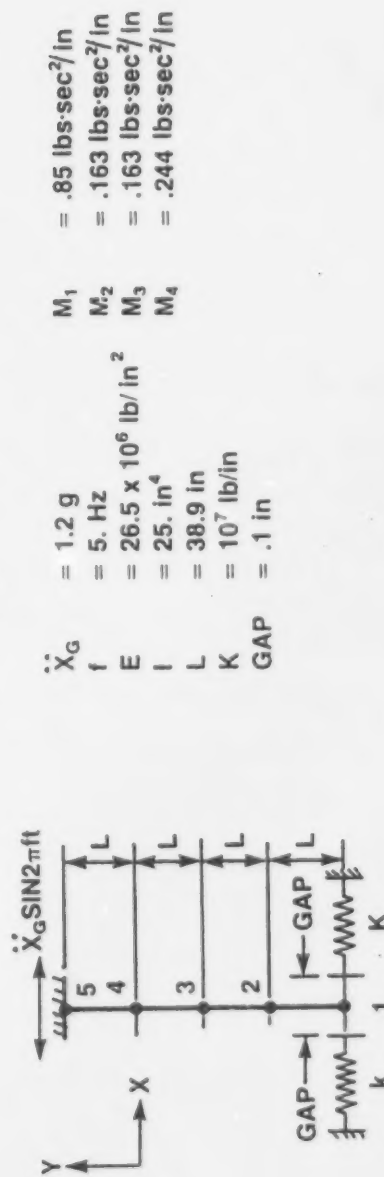
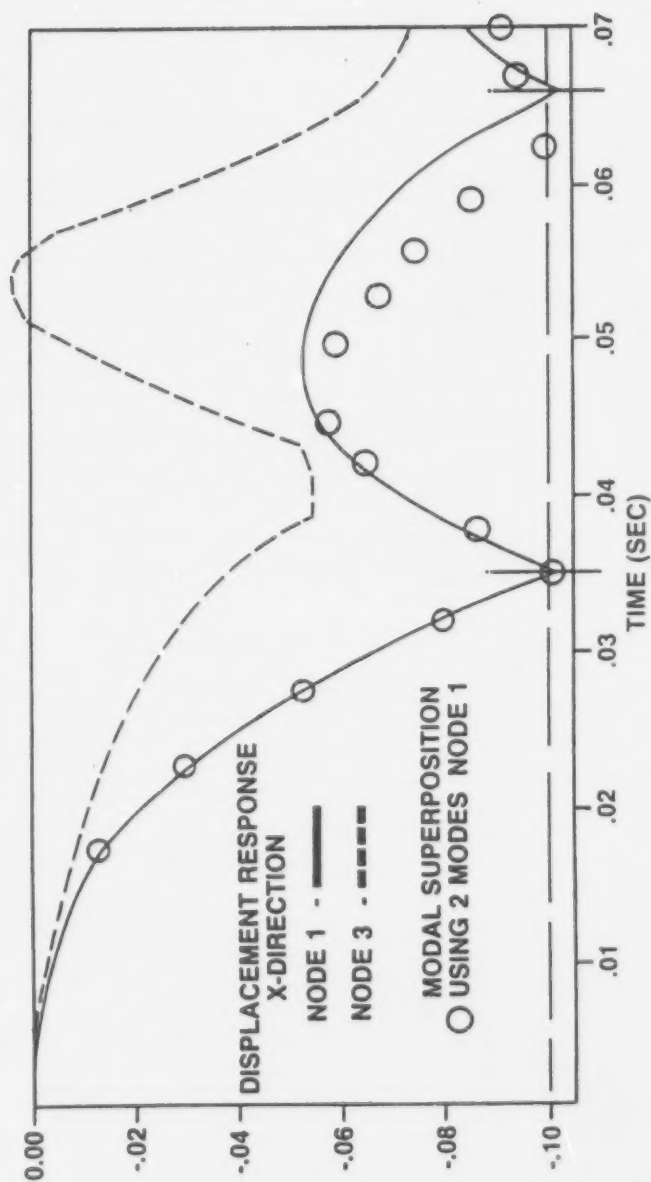


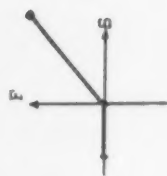
FIGURE 2.8.2



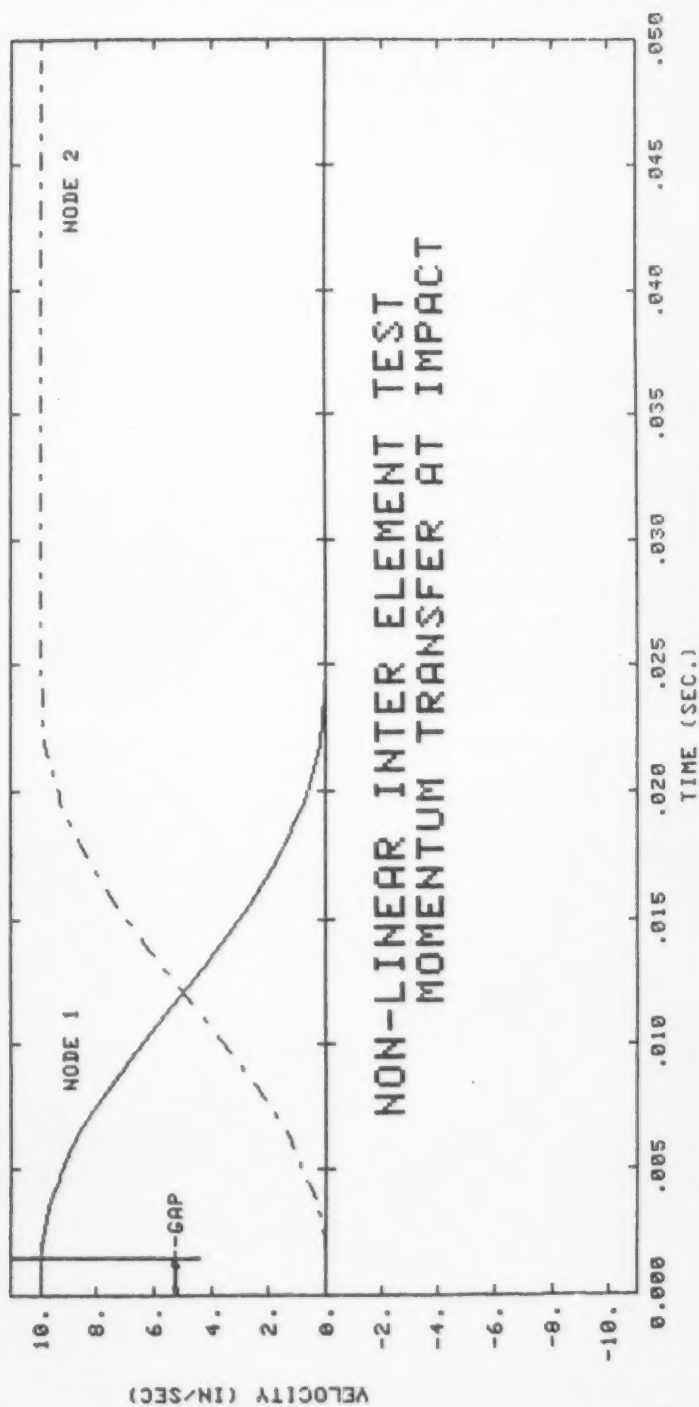
CASE 9: NON-LINEAR TRANSIENT DYNAMIC DISPLACEMENT RESPONSE

FIGURE 2.9.1

$V1 = 10 \text{ in/s}$
 $V2 = 0 \text{ in/s}$
 $M1 = 1.0 \text{ lb}$
 $M2 = 1.0 \text{ lb}$

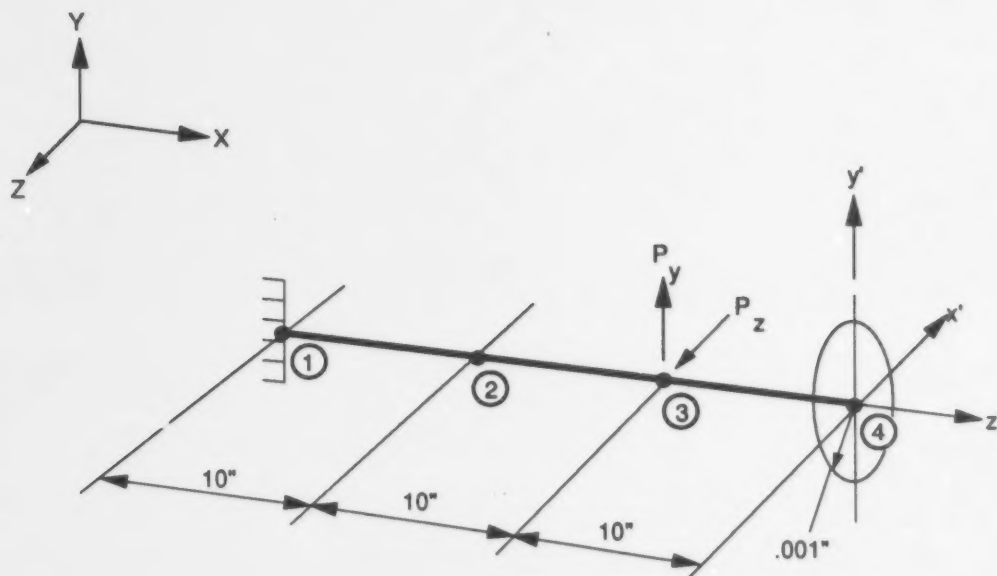


FORCE-DEFLECTION CURVE



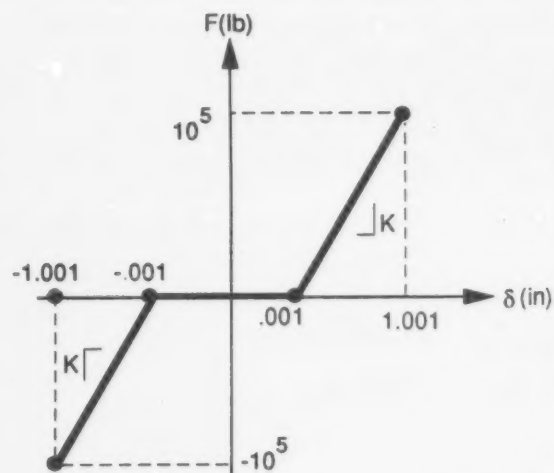
CASE 10: VELOCITY RESPONSE DURING MOMENTUM TRANSFER

FIGURE 2.10.1



$M2 = M3 = M4 = .1 \text{ lb sec}^2/\text{in}$
 $Iz' = Iy' = .5 \text{ in}^4$
 $A = 10.0 \text{ in}^2$
 $E = 10^6 \text{ lb/in}^2$
 $\mu = 0.3$
 $CDAMP = .01$

$K (\text{LINEAR}) = 1 \times 10^5 \text{ lb/in}$
 $P_y = 1.0 \sin \omega t$
 $P_z = 0.5 \sin (\omega t + 90^\circ)$



Force-Deflection Curve for the Annular Element

FIGURE 2.11.1

CASE 11: NON-LINEAR ANNULAR ELEMENT TEST MODEL

##.NON-LINEAR ANNULAR ELEMENT OPTIION TEST 2##

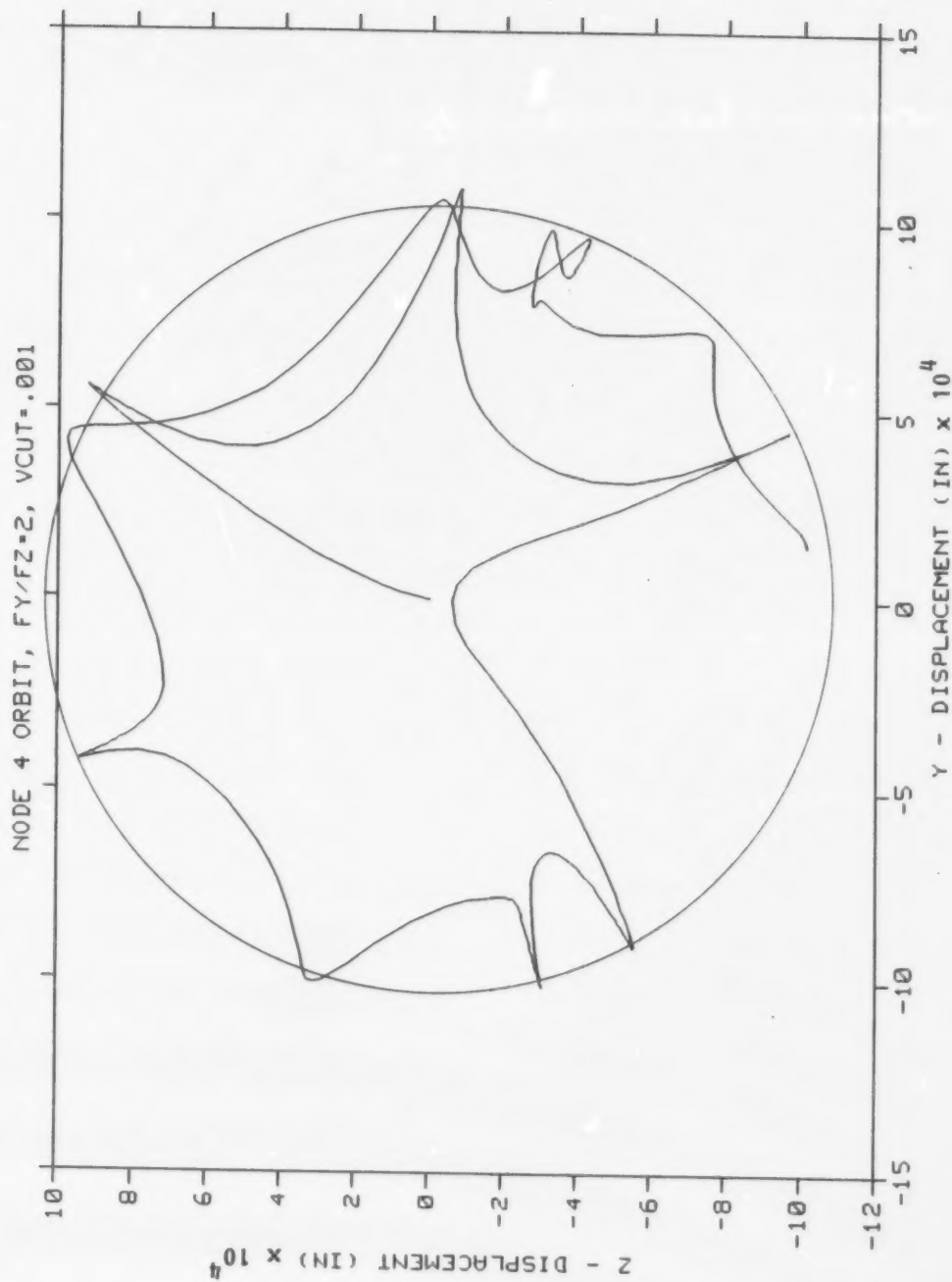
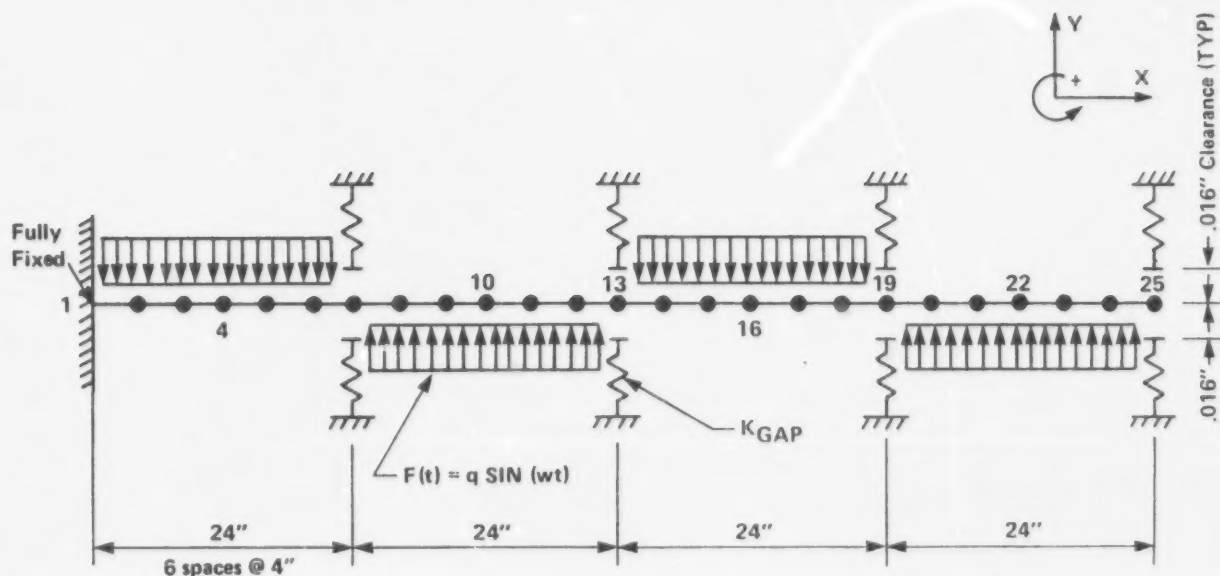


FIGURE 2.11.2

CASE 11: ORBIT OF NODE 4 IN AN ANNULUS

CASE 12: AECL CHALK RIVER/ONTARIO HYDRO
FRETTING FORCE PREDICTION
ONTARIO HYDRO BENCHMARK MODEL



DATA:

Tube: 0.625" OD, 0.045" Wall Thk.

Mat'l: E = Modulus of Elasticity = $30 \cdot 10^6$ psi

ρ = Density = .284 lb/in³

Non-Linear Elements: $K_{GAP} = 10^5$ lb/in (Linear Load Deflection Curve)
(at Nodes 7, 13, 19, 25) Radial Clearance = 0.016"

Forcing Function: Uniform Distribution on Each Span as Shown, Change in Direction at Each Span

$$F(t) = q \sin(wt)$$

$$q = 0.2 \text{ lb/in}$$

$$w = 2 \pi f$$

$$f = 40 \text{ Hz}$$

FIRST RUN:

No Damping (Internal or External)

SECOND RUN:

Support Damping = 0.28

Modal Damping Ratio = 0.005

Time Duration: .45 sec

Initial Conditions: $x = \dot{x} = \ddot{x} = 0$

FIGURE 2.12.1

***H3DMAP BENCHMARK WITH CRNL--TSP RADIAL CLEAR=.406 MM--**

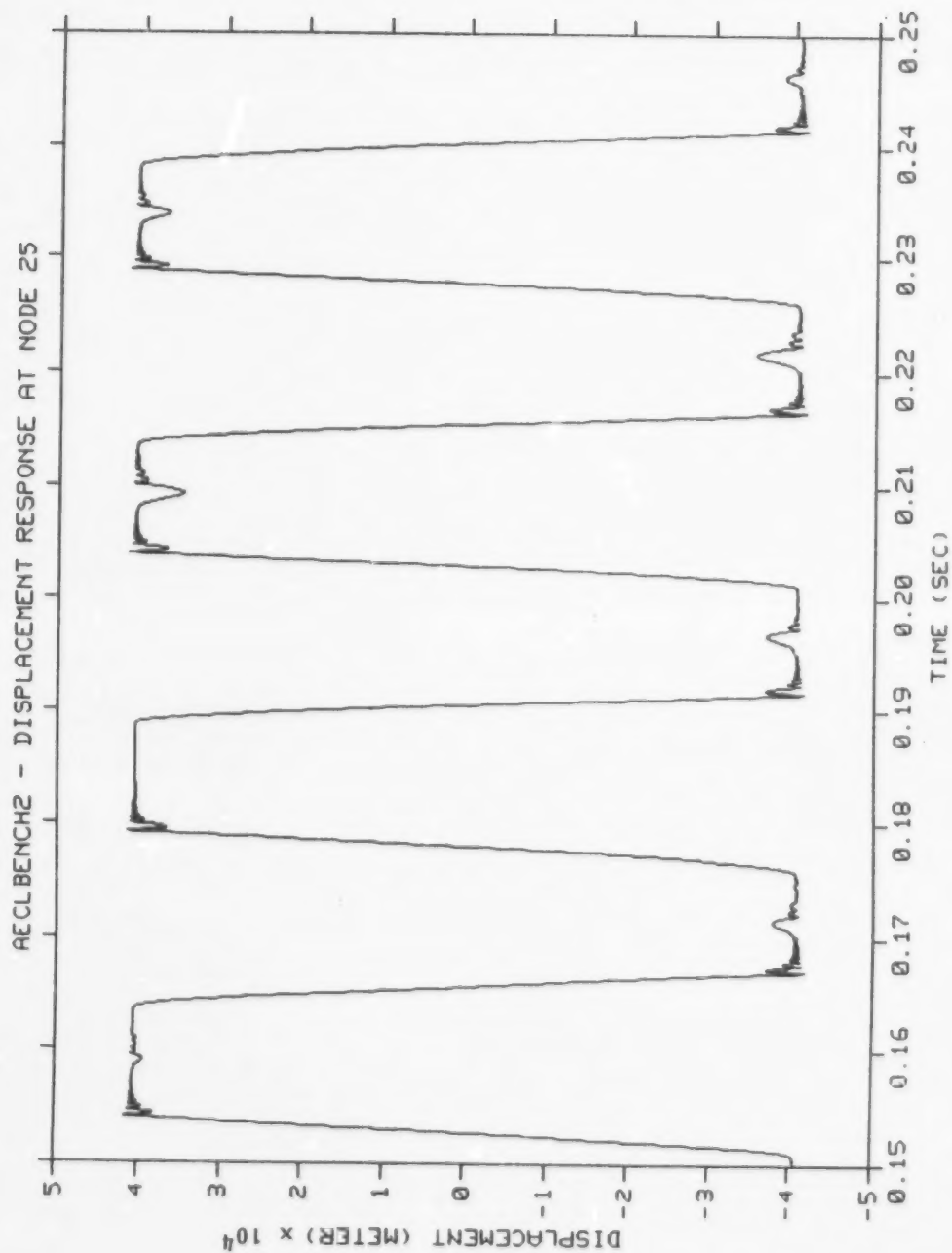


FIGURE 2.12.2

***H3DMAP BENCHMARK WITH CRNL--TSP RADIAL CLEAR=.406 MM--**

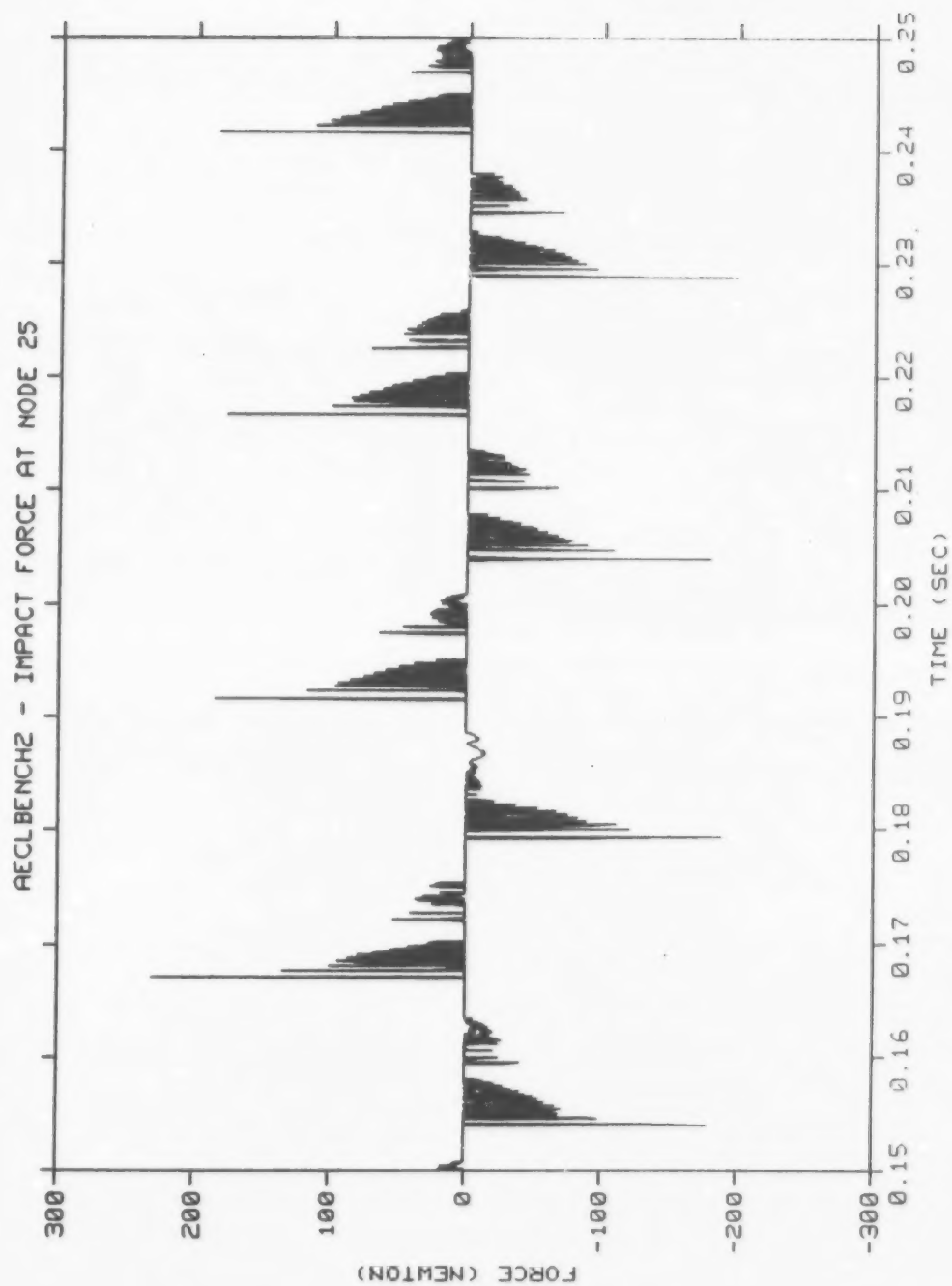


FIGURE 2.12.3

***H3DMAP BENCHMARK WITH CRNL--TSP RADIAL CLEAR=.406 MM--**

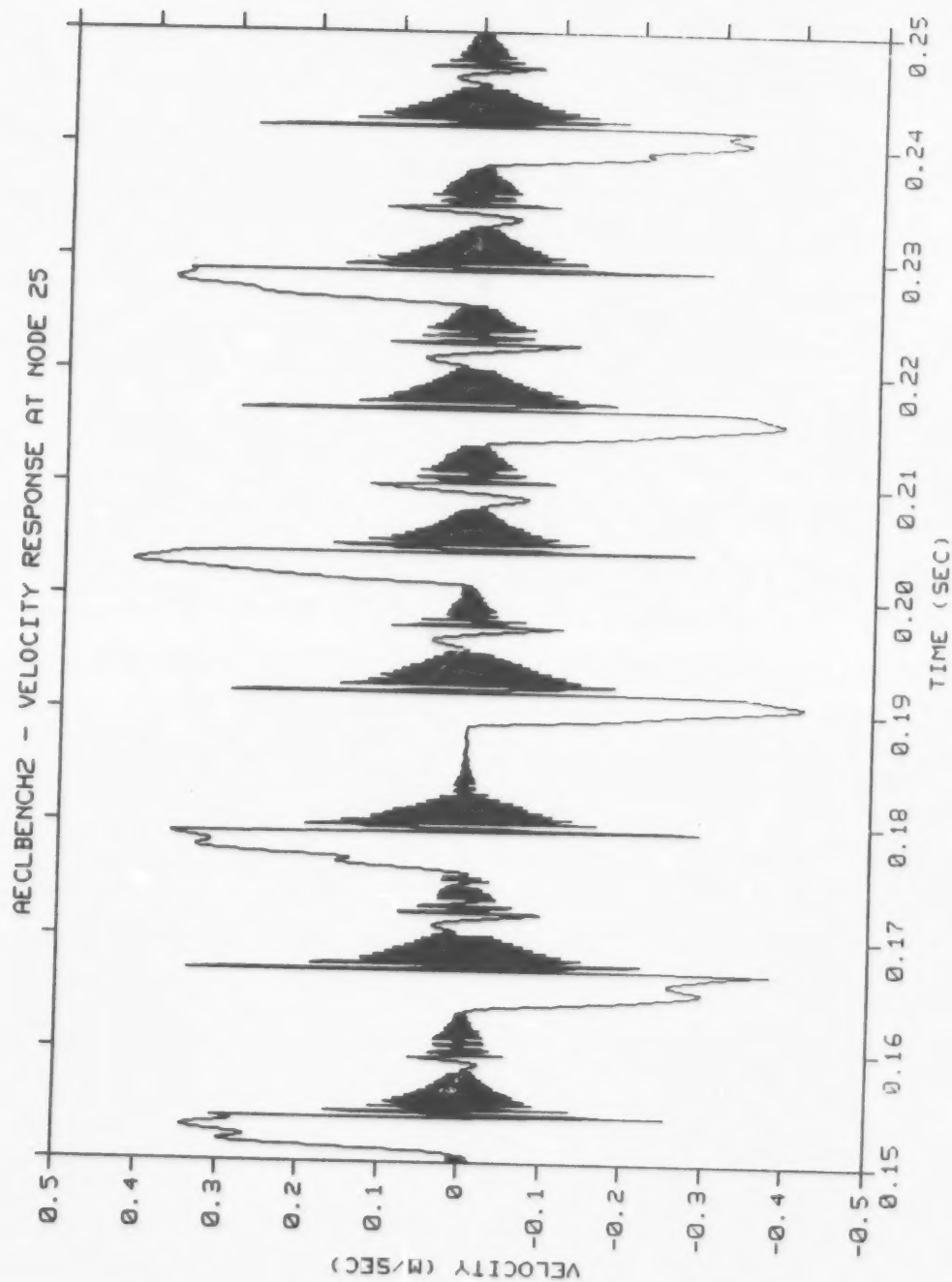


FIGURE 2.12.4

CASE 12: FORCE VS. TIME
SUPPORT NO. 4
CRNL BENCHMARK MODEL
RESULTS

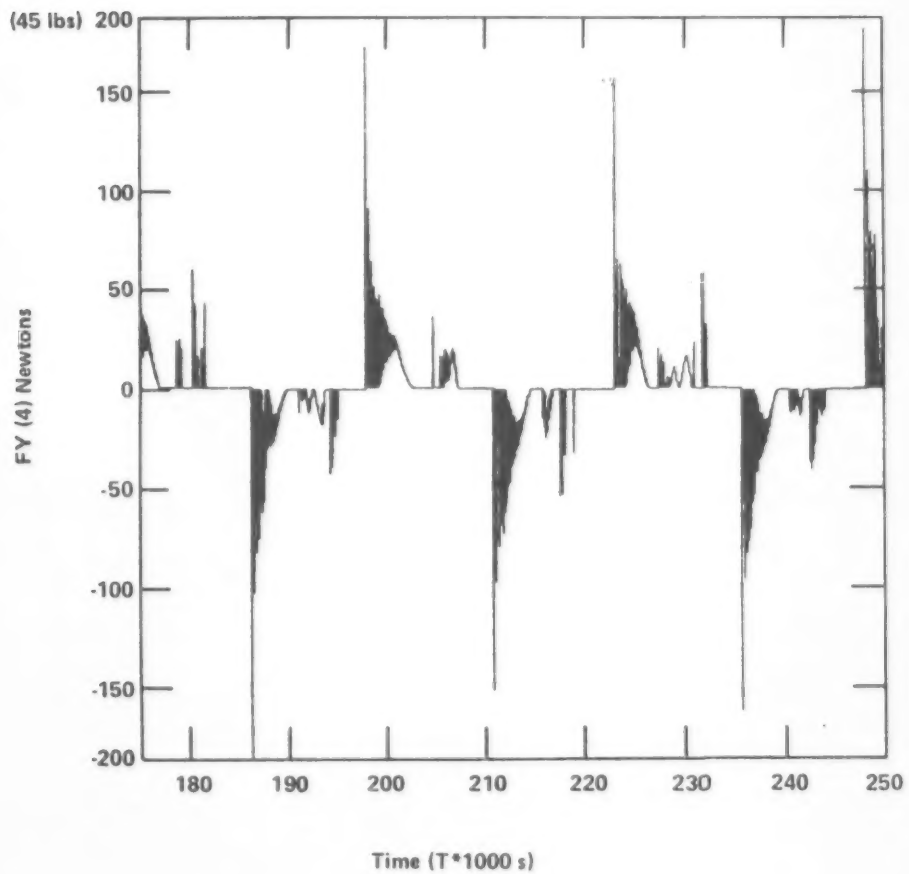
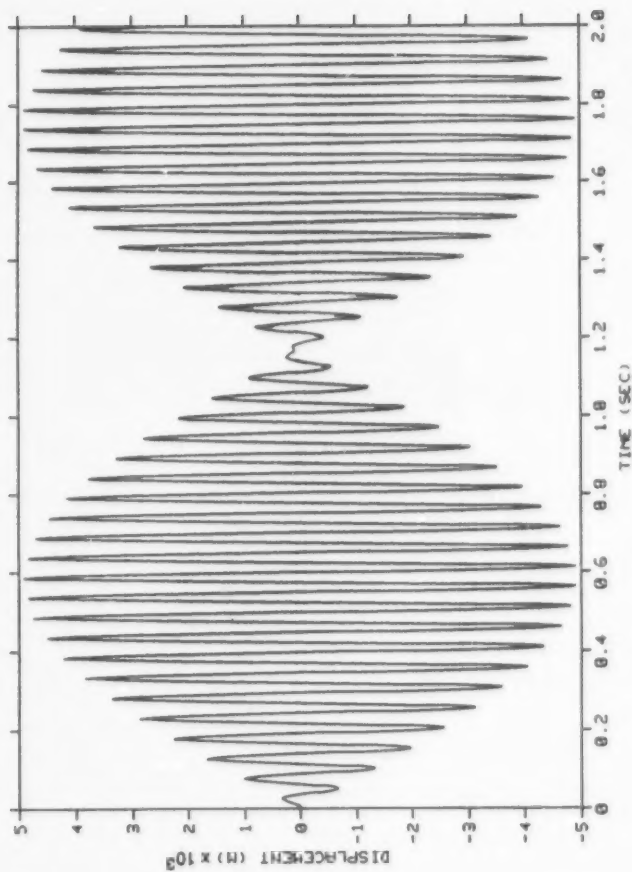


FIGURE 2.12.5



NODE 7

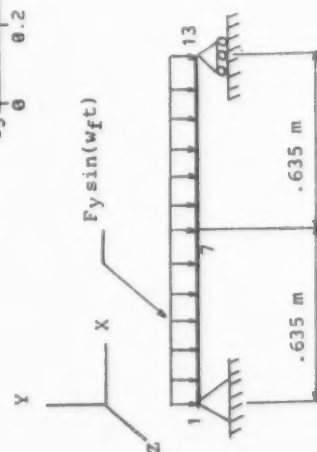
Undamped $f_1 = 19.3$ Hz

$f_f = 19.999$ Hz

Beat Frequency = $\frac{\Delta f}{2} = .35$ Hz

Calculated Beat Period = $\frac{1}{.35} = 2.85$ (based on first mode)

Predicted Beat Period = 2.6 (Combined modes)



$L = 1.27$ m

O.D. = 15.85 mm

I.D. = 13.61 mm

$E = 1.155 \times 10^{11}$ Pa

$\rho = 8.027 \times 10^{-3}$ kg/m³

$F_y = 1.0835$ N/m

$f = 20$ Hz

$w_f = 2\pi f$

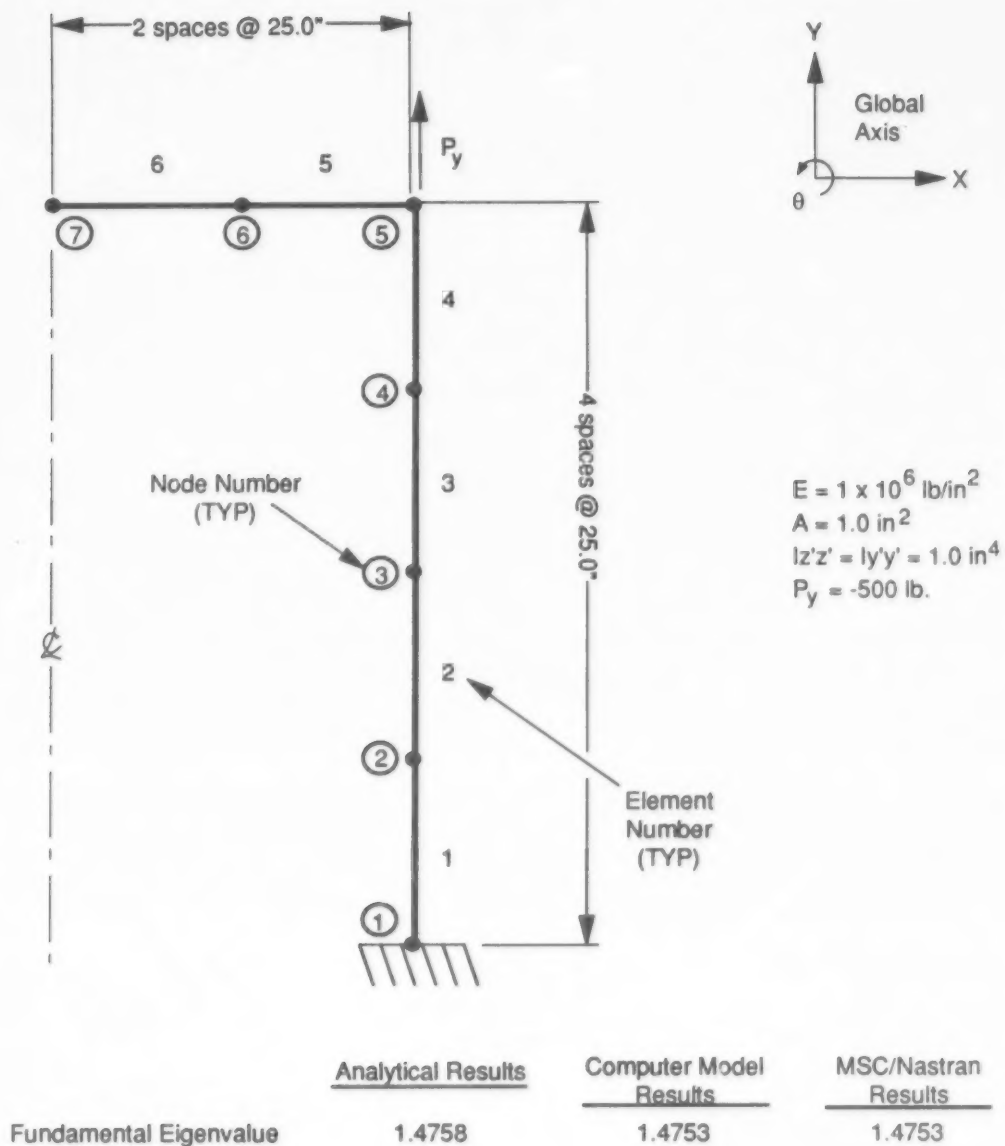
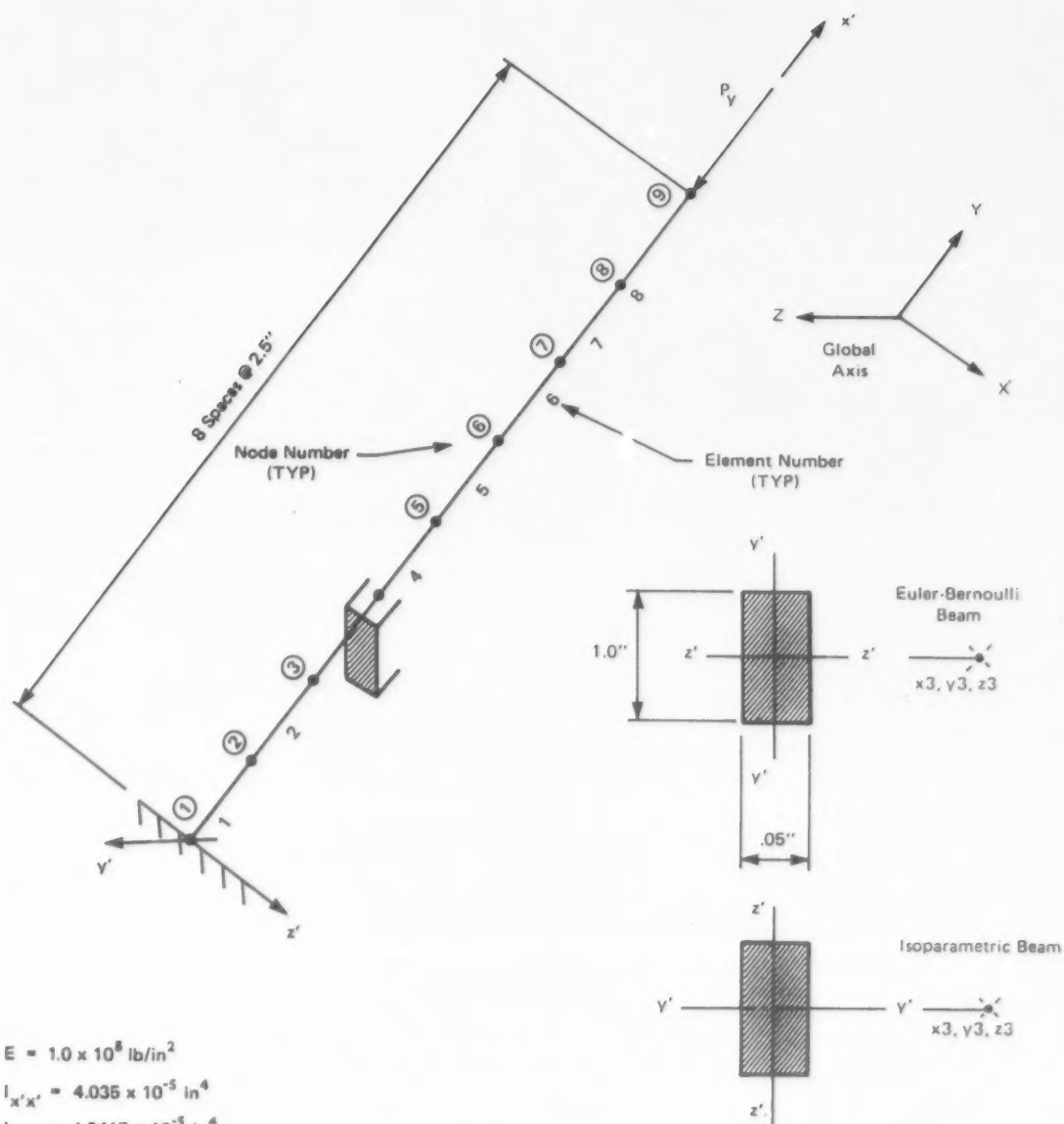


FIGURE 2.14.1

CASE 14.1: FRAME STABILITY ANALYSIS - COMPUTER MODEL



$$E = 1.0 \times 10^8 \text{ lb/in}^2$$

$$I_{x'x'} = 4.035 \times 10^{-5} \text{ in}^4$$

$$I_{y'y'} = 1.0417 \times 10^{-5} \text{ in}^4$$

$$I_{z'z'} = .00417 \text{ in}^4$$

$$P_y = 50 \text{ lb.}$$

Fundamental Eigenvalue

Euler-Bernoulli Beam
Formulation

0.12851

Linear Isoparametric Beam
Formulation

0.12934

Analytical
Results

0.1285

FIGURE 2.14.2
Case 14.2/33.4: Euler Column Buckling - Computer Model

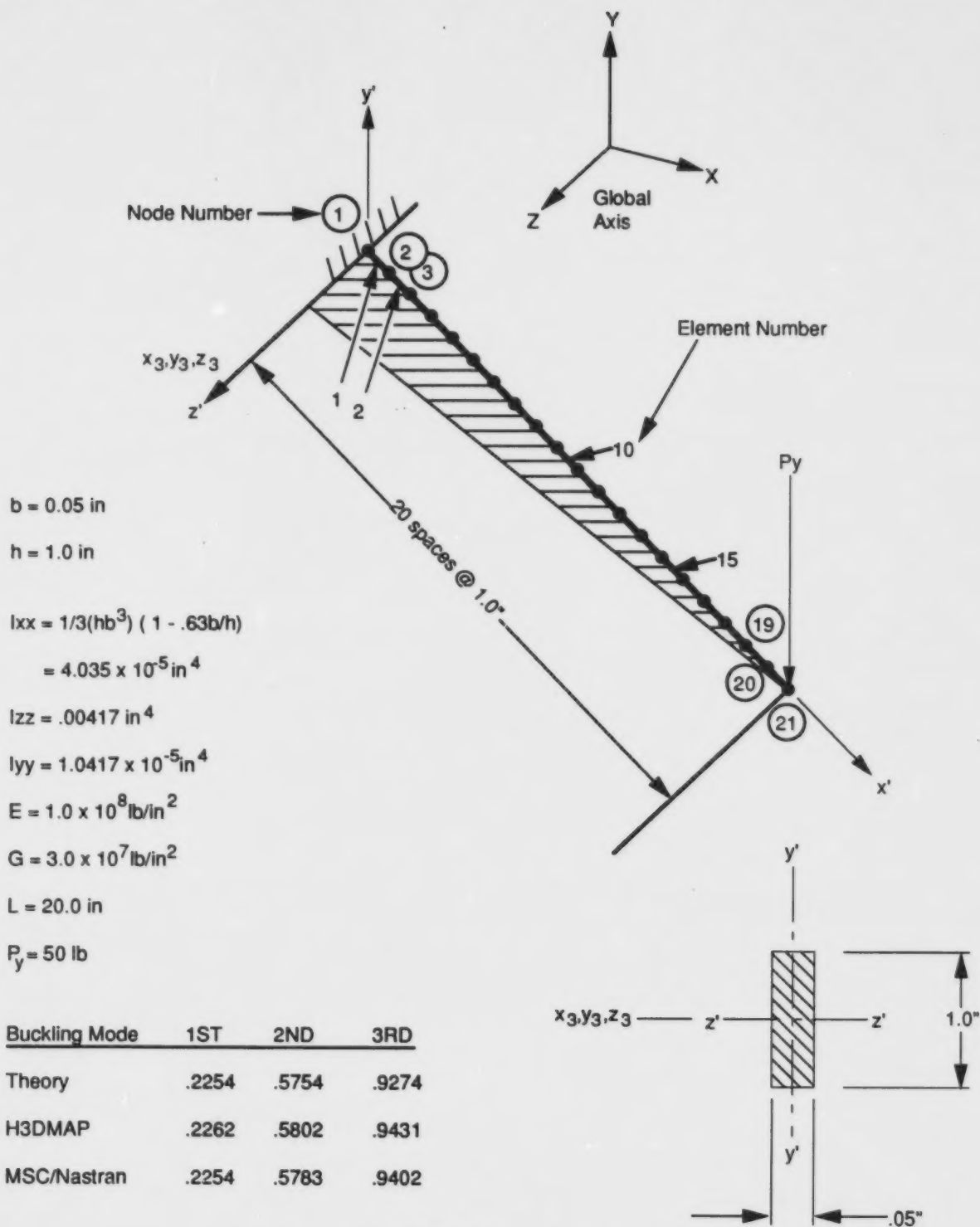
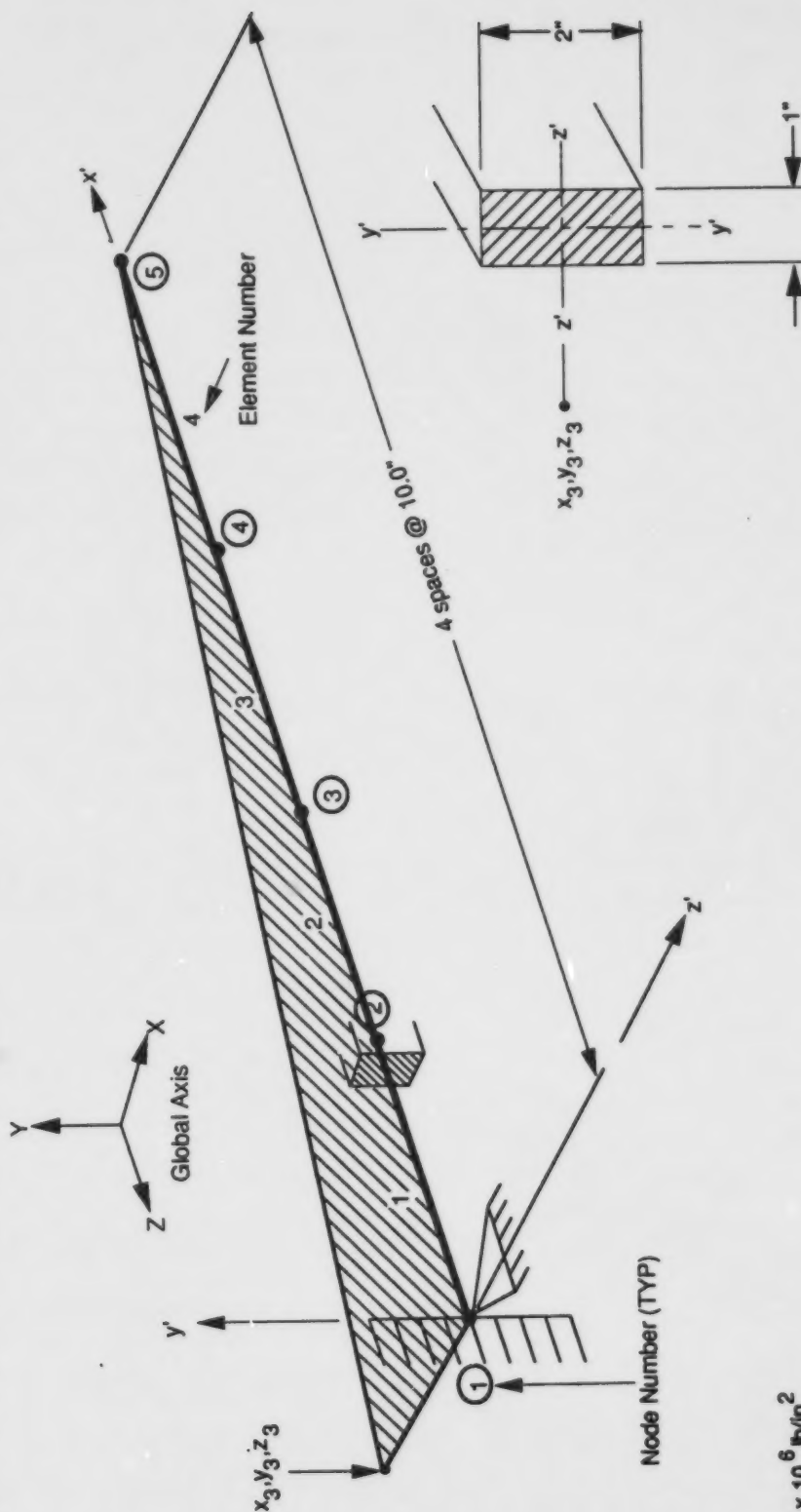


FIGURE 2.15.1

CASE 15: LATERAL BUCKLING ANALYSIS - COMPUTER MODEL



$$E = 30.0 \times 10^6 \text{ lb/in}^2$$

$$\alpha = .65 \times 10^{-5} \frac{\text{in}}{\text{in } ^\circ\text{F}}$$

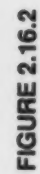
$$T_0 = 70.0 ^\circ\text{F}$$

$$T_1 = 170.0 ^\circ\text{F}$$

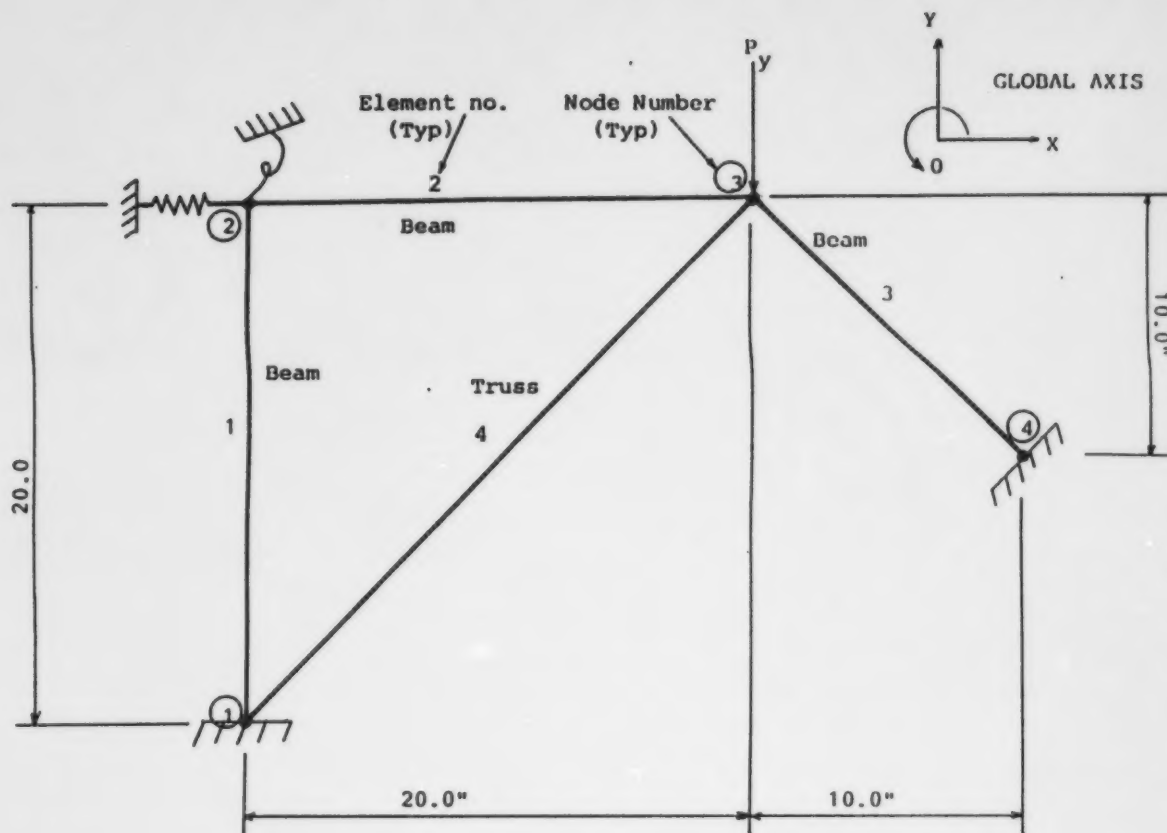
Node Number 5	Analytical Results	Computer Model Results
Global Z - Displacement (in)	.026	.026

FIGURE 2.16.1

CASE 16.1: STRAIGHT BEAM THERMAL GROWTH - COMPUTER MODEL



CASE 16.2: STRAIGHT BEAM THERMAL LOAD - COMPUTER MODEL



$$E_B = 1000. \text{ lb/in}^2$$

$$EI_B = 10000. \text{ lb-in}^2$$

$$A_B = 100. \text{ in}^2$$

$$E_T = 1000. \text{ lb/in}^2$$

$$EI_T = 50000. \text{ lb-in}^2$$

$$A_T = 50. \text{ in}^2$$

$$K_{x2} = 100 \text{ lb/in}$$

$$K_{02} = 1000 \text{ lb/in/rad}$$

$$P_y = 1000 \text{ lb}$$

$$T_1 = T_2 = T_3 = 70^\circ\text{F}$$

$$T_4 = 270.0^\circ\text{F}$$

$$\alpha = 2.0 \times 10^{-4} \frac{\text{in}}{\text{in}^\circ\text{F}}$$

	Analytical Results		Computer Model Results	
	Node 2	Node 3	Node 2	Node 3
Global X - Displacement (in)	-.3822	-.3909	-.3827	-.3914
Global Y - Displacement (in)	-.0011	-.1377	-.00107	-.1382
Global R _z - Displacement (in)	-.0037	-.0182	-.00372	-.0182

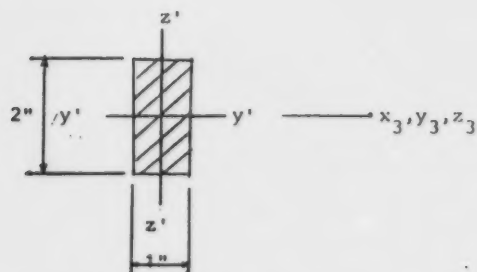
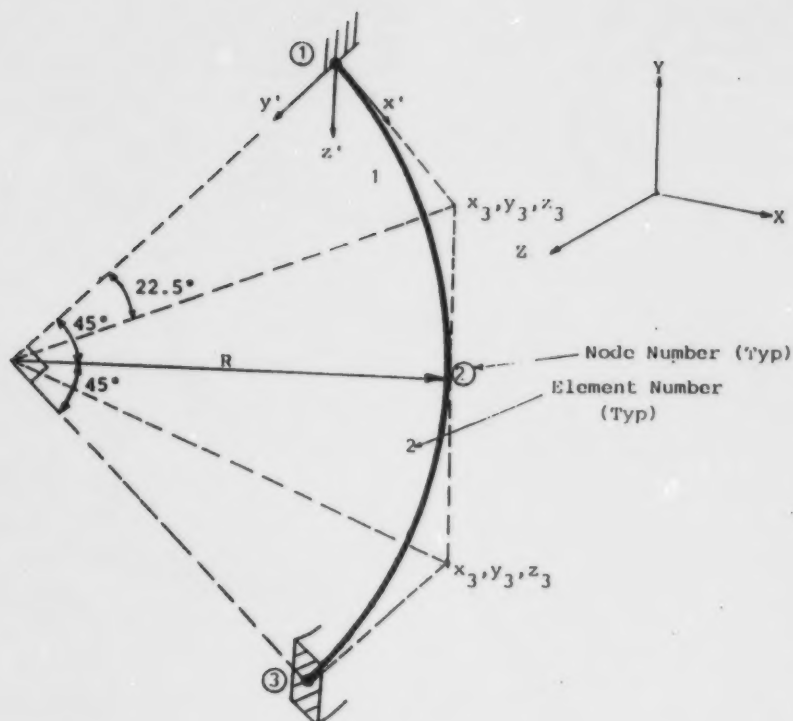
Element Number	1	2	3	4	
Local Axial Force (lb)	5.50	43.5	734.3	660.74	Analytical Results
Local Y-Axis Shear (lb)	-5.178	5.33	-10.389		
Max. Local Z-Axis Moment (lb-in)	-53.63	60.59	-86.4		
	1	2	3	4	
Local Axial Force (lb)	5.35	43.46	734.1	662.2	Computer Model Results
Local Y-Axis Shear (lb)	-5.178	5.33	-10.42		
Max. Local Z-Axis Moment (lb-in)	-53.69	60.77	-86.57		

CASE 17: MIXED FRAME TEST - COMPUTER MODEL

FIGURE 2.17.1

181026-RD

$$\begin{aligned}
 R &= 100 \text{ in} \\
 E &= 30 \times 10^6 \text{ lb/in}^2 \\
 T_0 &= 70^\circ\text{F} \\
 T_1 &= 170^\circ\text{F} \\
 \alpha &= .65 \times 10^{-5} \frac{\text{in}}{\text{in}^\circ\text{F}}
 \end{aligned}$$

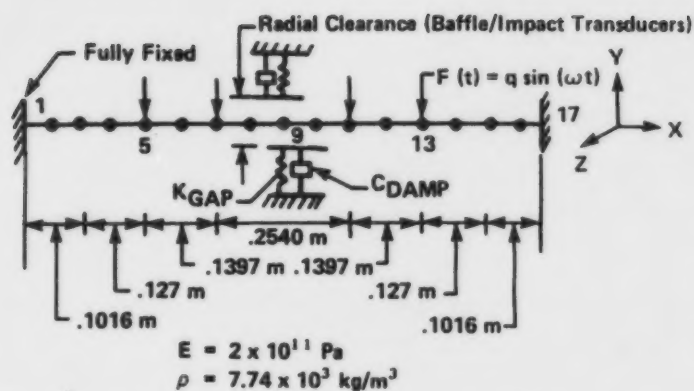


Node 3	Analytical Results	Computer Model Results
Global Z-DIRECTION DISPLACEMENT (IN)	.0650	.065004
Global X-DIRECTION DISPLACEMENT (IN)	.0650	.064992

CASE 18: CURVED BEAM THERMAL GROWTH - COMPUTER MODEL

FIGURE 2. B.1

181027-RD



Tube: .9906 m LG
 15.375 mm OD
 9.525 mm ID

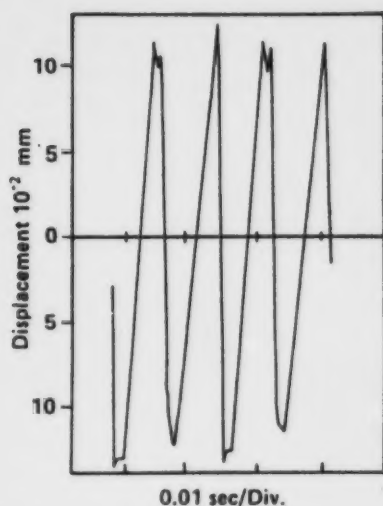
Support: Stiffness = $K_{GAP} = 1.75 \times 10^6 \text{ N/m}$
 Material Constant $C_{DAMP} = 0.2 \text{ N-s/m}$
 Radial Clearance = 0.127 mm

Tube $C = \beta M + \gamma K$,
 Damping: $\beta = 4.0$,
 $\gamma = 5.0 \times 10^{-6}$

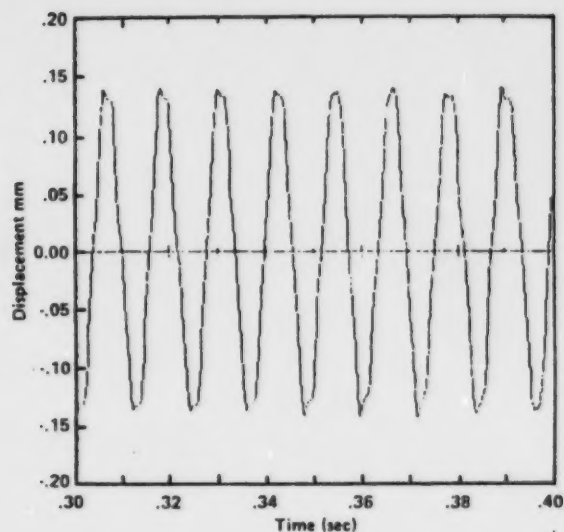
Forcing $q = 2.95 \text{ N}$
 Function: $f = 84 \text{ Hz}$, $\omega = 2\pi f = 527.788 \text{ rad/s}$
 $F(t) = 2.85 \sin(527.788 t)$

FIGURE 2.19.1
 CASE 19: EXPERIMENTAL BENCHMARK MODEL (REF. 29)

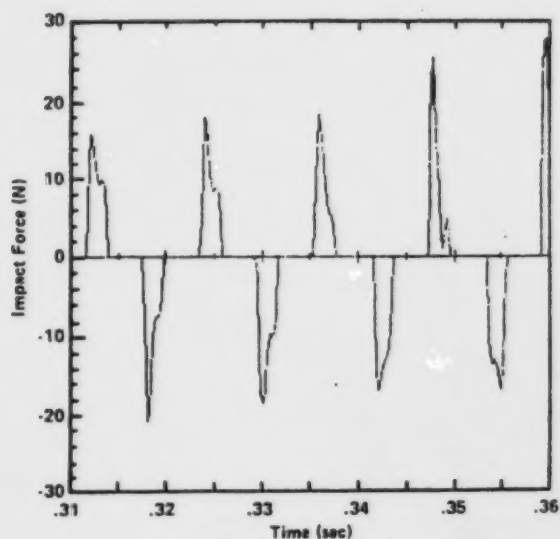
MEASURED DISPLACEMENT



PREDICTED DISPLACEMENT (NODE 9)



PREDICTED IMPACT FORCE



MEASURED IMPACT FORCE



CASE 19: MEASURED/PREDICTED DISPLACEMENT AND IMPACT FORCE RESPONSE

FIGURE 2.19.2

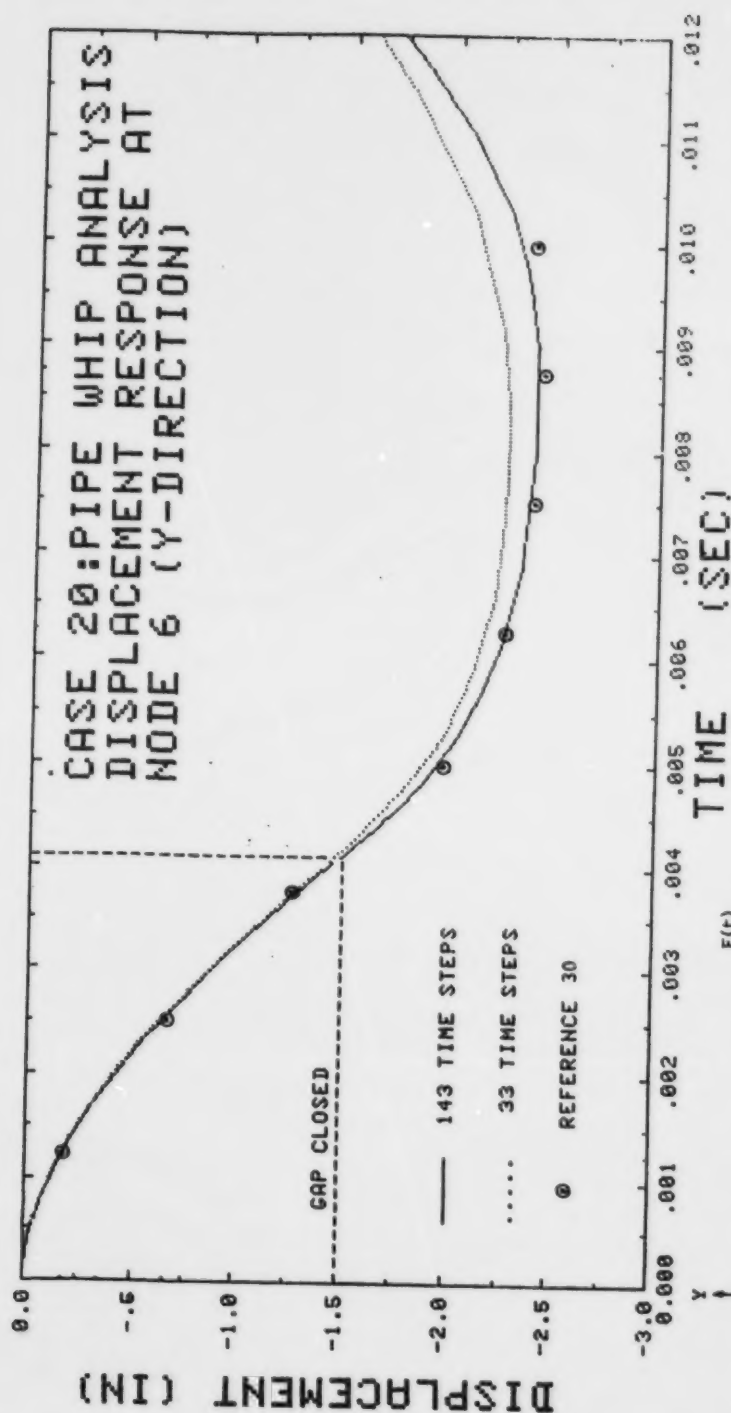


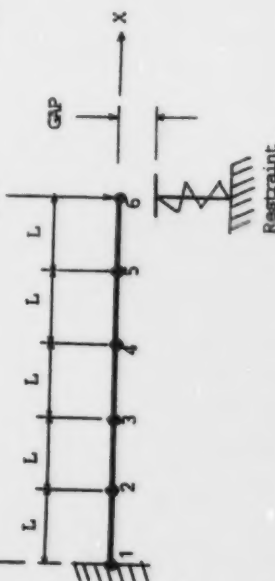
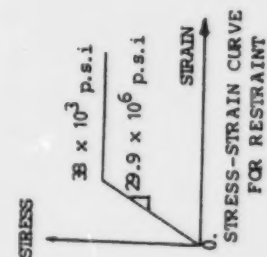
FIGURE 2.20.1

Data
 $F(t)$ = constant blow down force
= 6.57×10^5 lbs

Beam
 L = 72 in
 E = 26.9×10^6 p.s.i
 I = 10652 in^4
 A = $102. \text{ in}^2$
Density = $.23 \text{ lb/in}^3$

Restraint

GAP = 1.5 in Area = 25.97 in^2 Length = 24 in



181030-RD

DISPLACEMENT (IN.)

TIME (SEC)

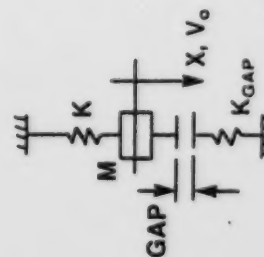
TIME STEP (sec)

VARIATION OF TIME STEP

DISPLACEMENT RESPONSE FOR SINGLE DEGREE OF FREEDOM MODEL

X_{MAX}

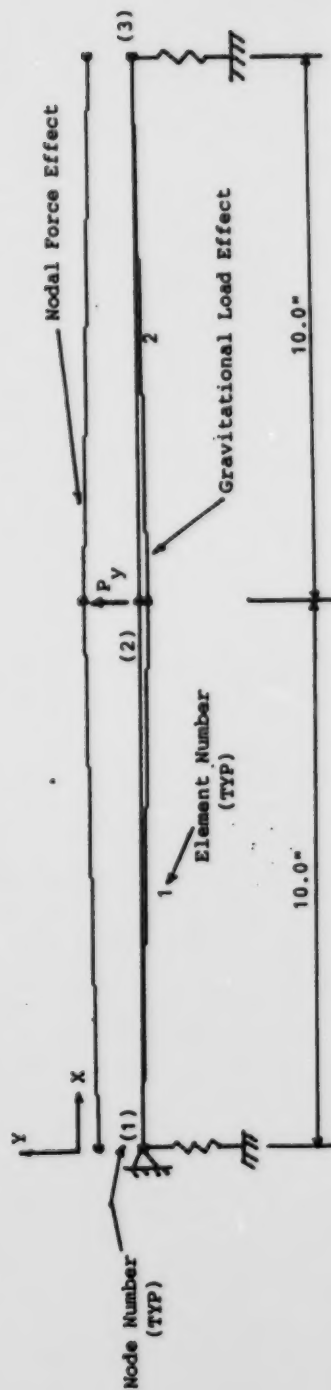
GAP CLOSED

 $V_o = 10 \text{ in/sec (initial velocity)}$

$K = 400 \text{ lb/in}$
 $M = 100 \text{ lb-sec}^2/\text{in}$
 $K_{\text{GAP}} = 10^5 \text{ lb/in}$
 $\text{GAP} = 1. \text{ in}$

CASE 21: Single Degree of Freedom Model - Displacement Response

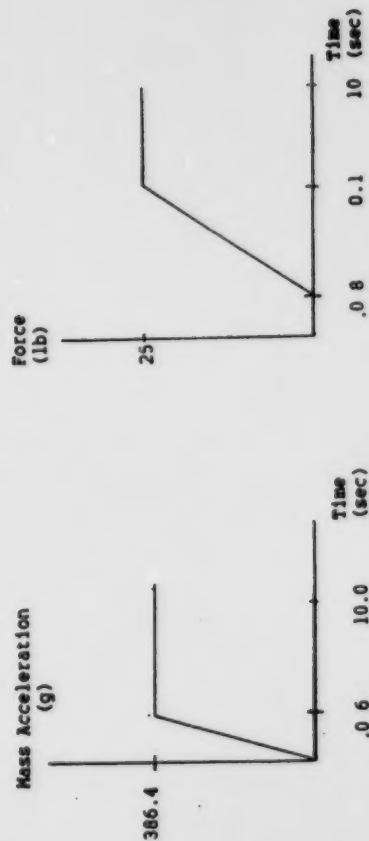
Figure 2.21.1



P_y	=	25 lb
E	=	Young's Modulus = 1.0×10^6 psi
I	=	Moment of Inertia = 1.0 in^4
ρ	=	Density = 0.1 lb/in^3
A	=	Area = 10.0 in^2
β	=	Mass Damping Coefficient = 1000.0

Non-linear Elements: $K = 10^6 \frac{\text{lb}}{\text{in}}$

(at nodes 1 and 3) (Linear - Load Deflection Curve)



181032-RD

CASE 22: DYNAMIC RELAXATION COMPUTER MODEL AND RESPONSE

FIGURE 2.22.1

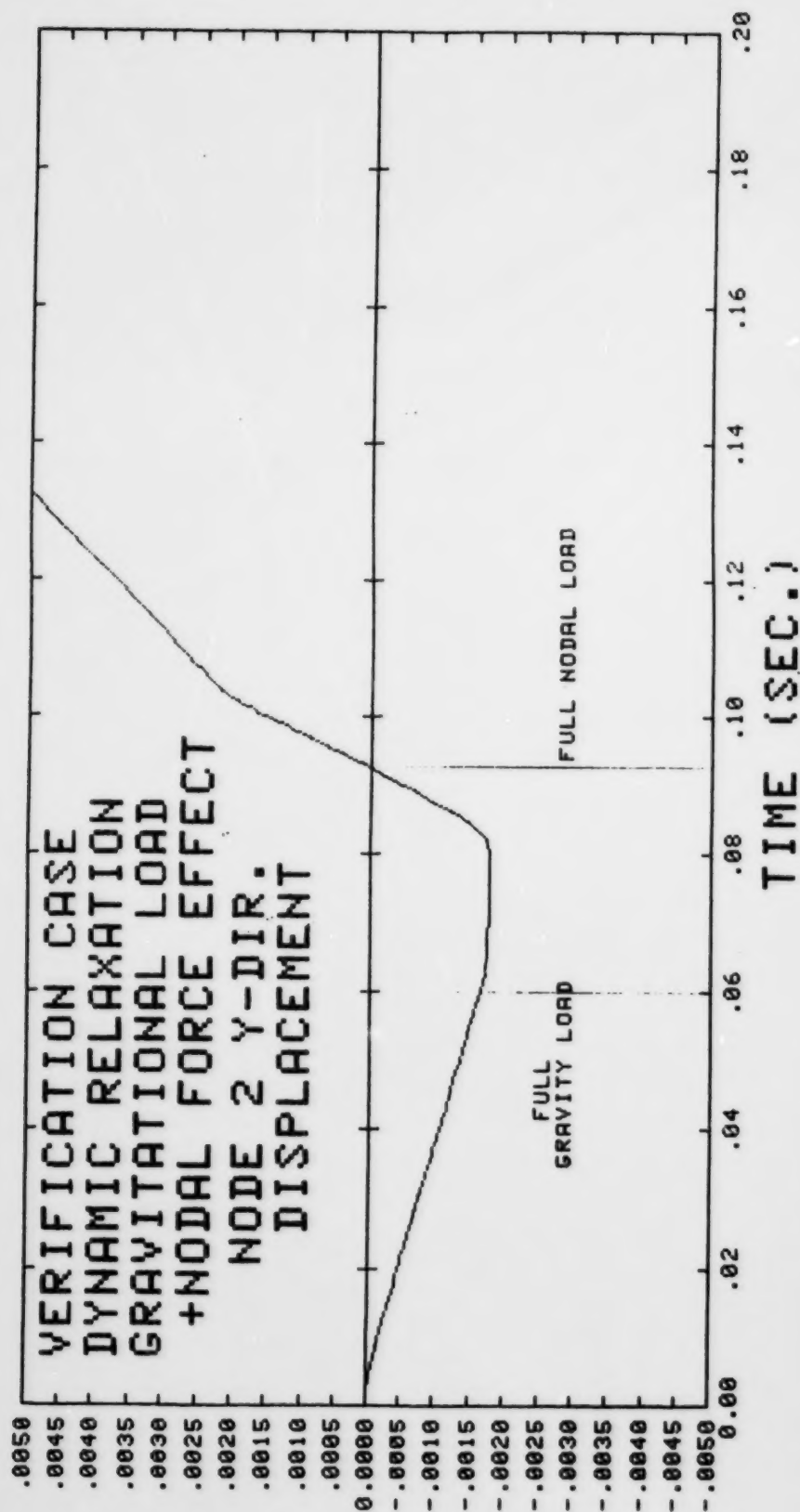
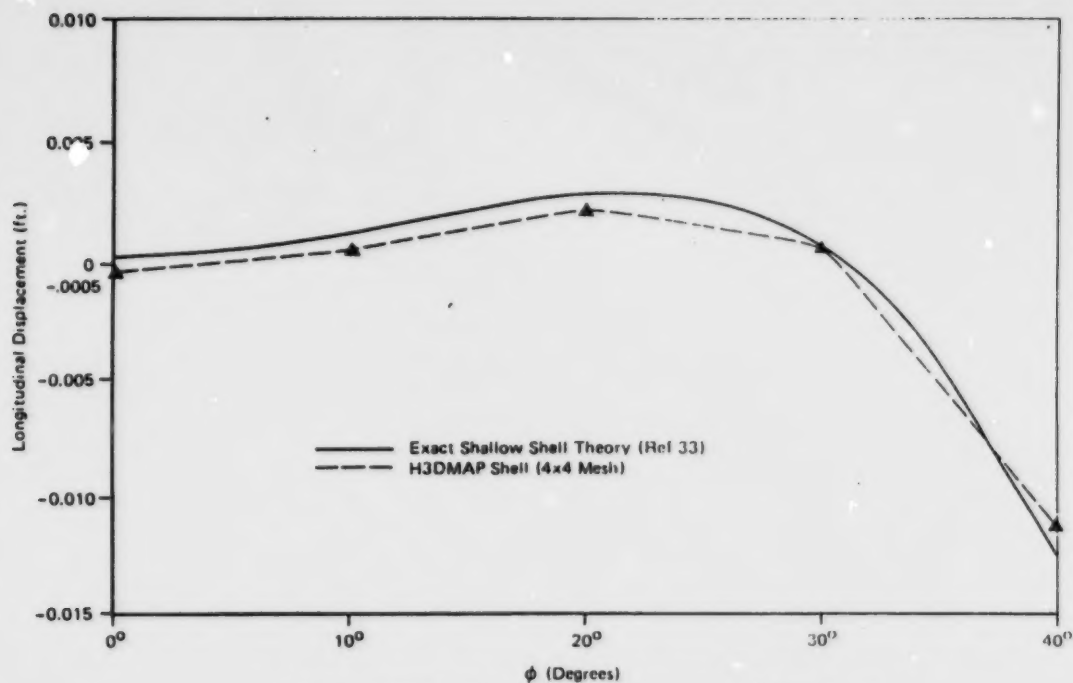
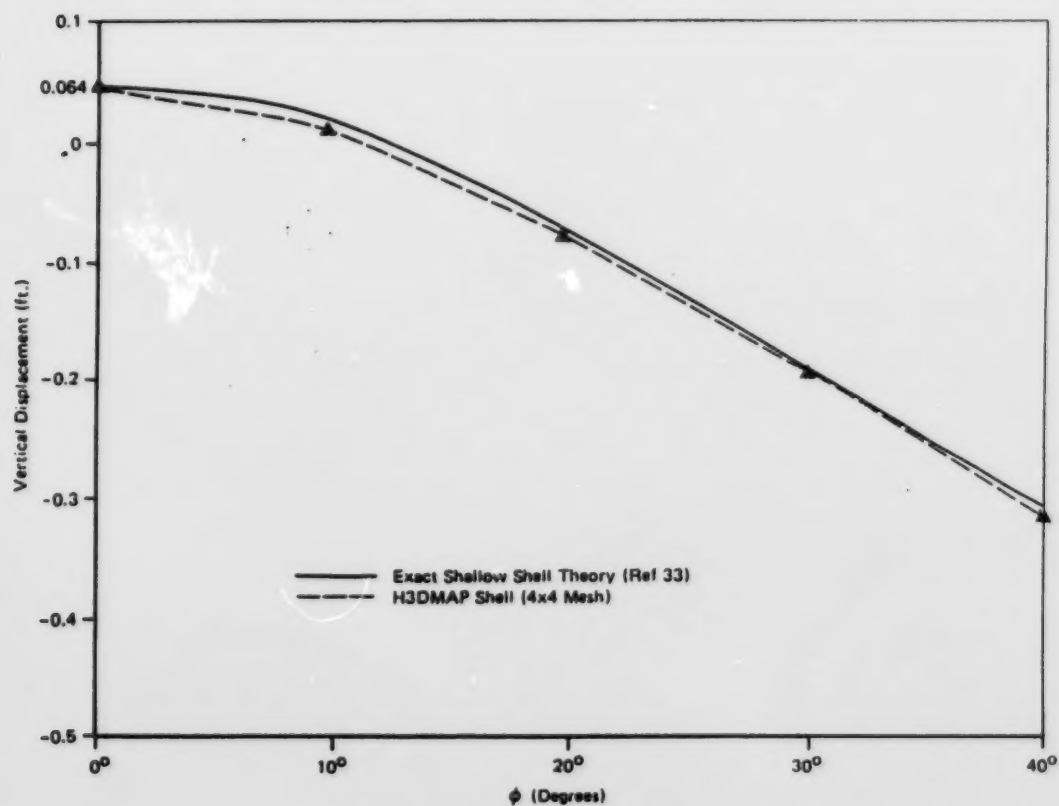


FIGURE 2.22.2

CASE 22: DISPLACEMENT TIME HISTORY CURVE AT NODE 2



Cylindrical Shell Roof, Longitudinal Displacement at Diaphragm ($y = 0$)

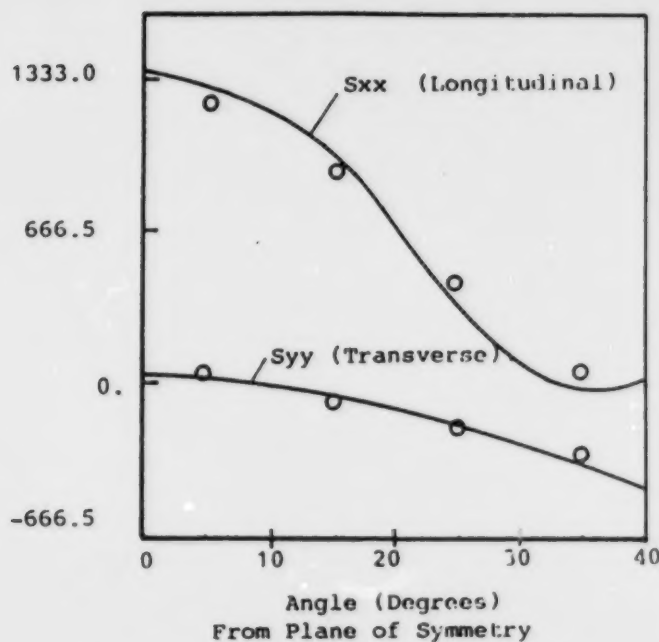


Cylindrical Shell Roof, Vertical Displacement at $y = L/2$

FIGURE 2.23.2
Case 23 - Thin Cylindrical Shell Roof
Longitudinal and Vertical Static Displacements

181035-RD

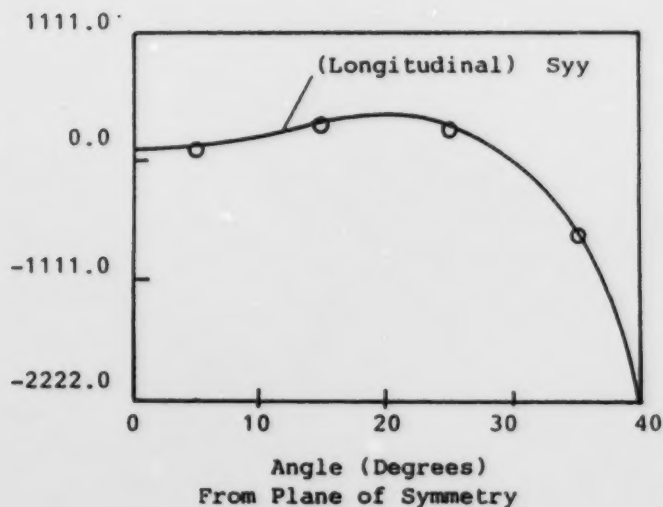
Local Sxx, Syy Stresses (psi)



— Exact Solution (Ref.33)
o H3DMAP Solution

BENDING STRESSES Sxx, Syy AT
CENTRAL SECTION (y=L/2)

Local Syy Stress (psi)



— Exact Solution (Ref. 33)
o H3DMAP Solution

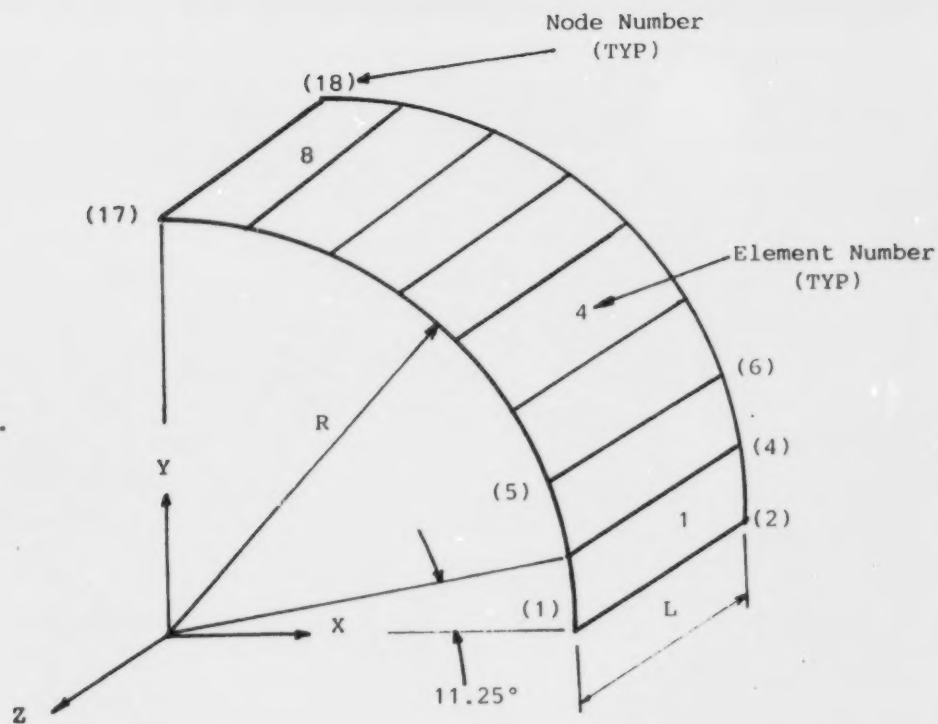
MEMBRANE STRESS Syy, At
CENTRAL SECTION (y=L/2)

CASE 23 - THIN CYLINDRICAL SHELL ROOF
BENDING/MEMBRANE STRESSES

FIGURE 2.23.3

181036-RD

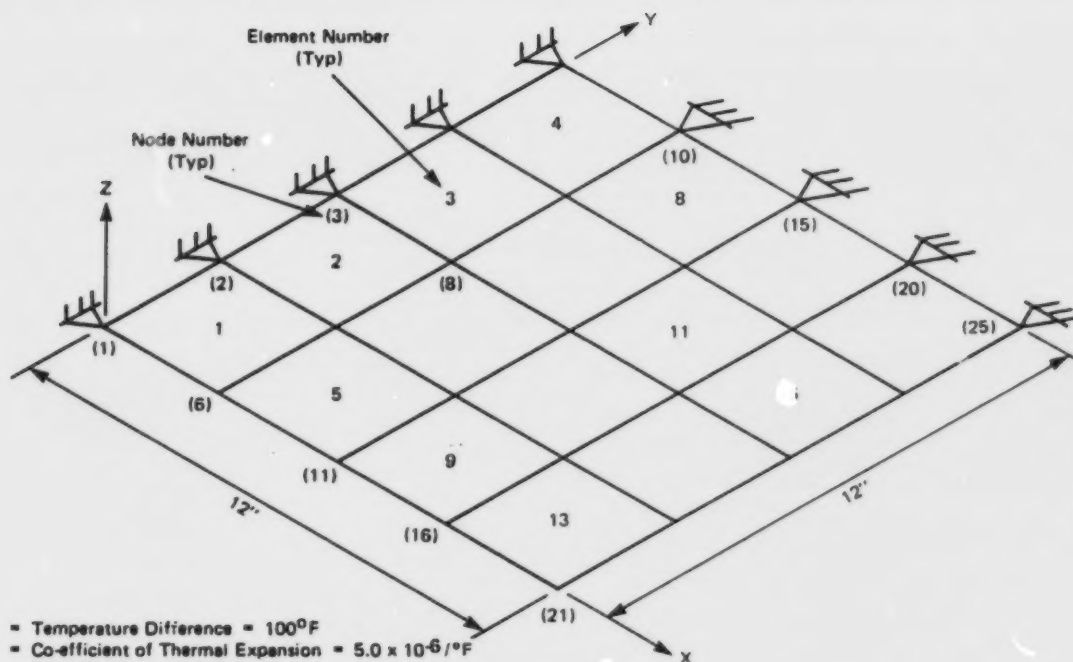
$P = 1000 \text{ p.s.i.}$
 $t = 0.1 \text{ inches}$
 $L = 1.0 \text{ inches}$
 $R = 10.0 \text{ inches}$
 $E = 30.0 \times 10^6 \text{ p.s.i.}$



	<u>H3DMAP RESULTS</u>		<u>Classical Theory</u>
	1x4 mesh	1x8 mesh	
Displacement (max.) (in)	0.02975	0.0302	0.02833
Stress (lb/in ²)	98080	99520	100,000

FIGURE 2.24.1

Thin Cylinder Under Internal Pressure - Computer Model



ΔT = Temperature Difference = $100^{\circ}F$
 ϕ = Co-efficient of Thermal Expansion = $5.0 \times 10^{-6}/^{\circ}F$
 t = Plate Thickness = 0.25 inches
 E = Young's Modulus = 30×10^6 p.s.i.
 μ = Poissons Ratio = 0.30

Loading Condition	Mesh Size	Node Number	Finite Element Solution			Classical Theory	
			Deflection (in)	Element Number	Centroidal Stress (psi)	Deflection (in)	Stress (psi)
Concentrated Load at Plate Centre	2 x 2	9	0.0832	--	--	0.0778	--
	4 x 4	21	0.0786	--	--		
Uniform Pressure Load	2 x 2	9	0.3328	4	23 990	0.3138	26487
	4 x 4	21	0.318	13	25 830		
Thermal Gradient	4 x 4	--	--	13	10 710	--	10714
In-Plane Expansion	4 x 4	25	0.0060	--	--	0.0060	--
Natural Frequencies	Full Model	Mode Number		Finite Element Solution Frequencies (Hz)		Classical Theory Frequencies (Hz)	
		1		82.01		83.33	
		2		208.2		208.33	
		3		208.2		208.33	
		4		322.7		333.46	
		5		431.6		416.71	

FIGURE 2.25.1
 Case 25: Simply Supported Square Plate Analysis

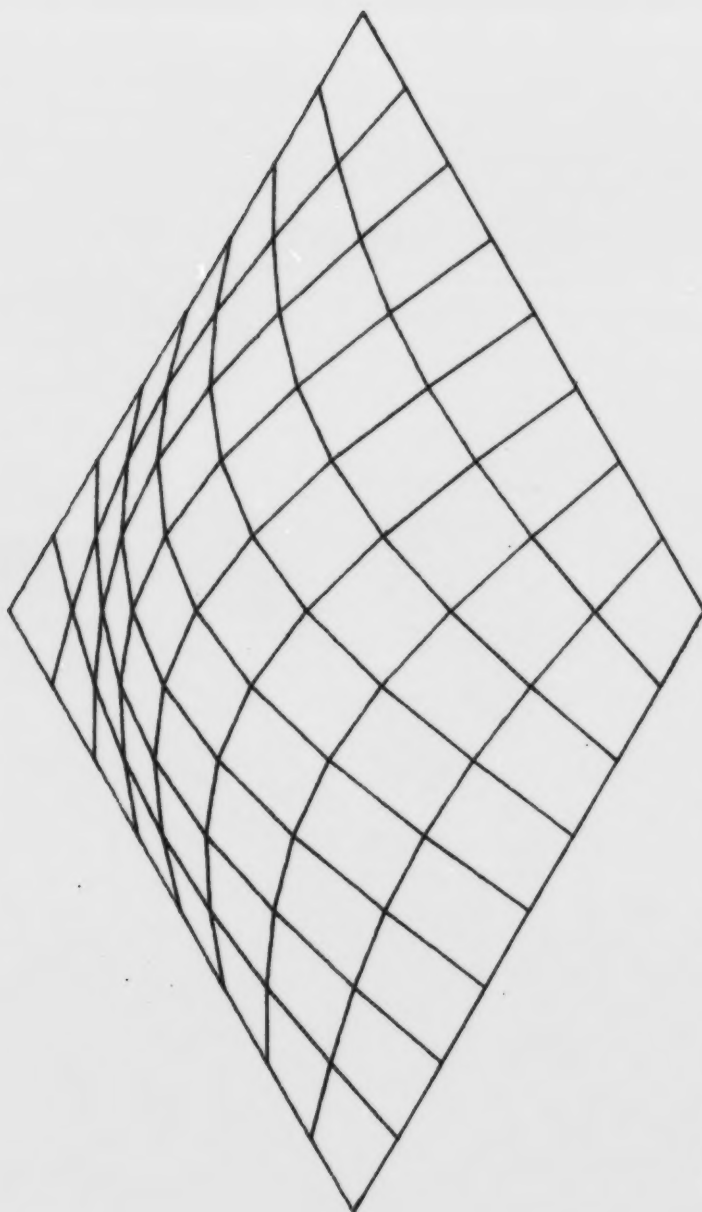
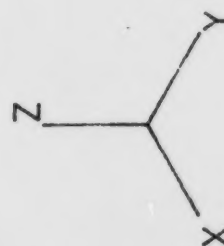


FIGURE 2.25.2

SIMPLY SUPPORTED SQUARE PLATE MODAL ANALYSIS

VIEW POINT 1.0 1.0 1.0 MODE SHAPE 1



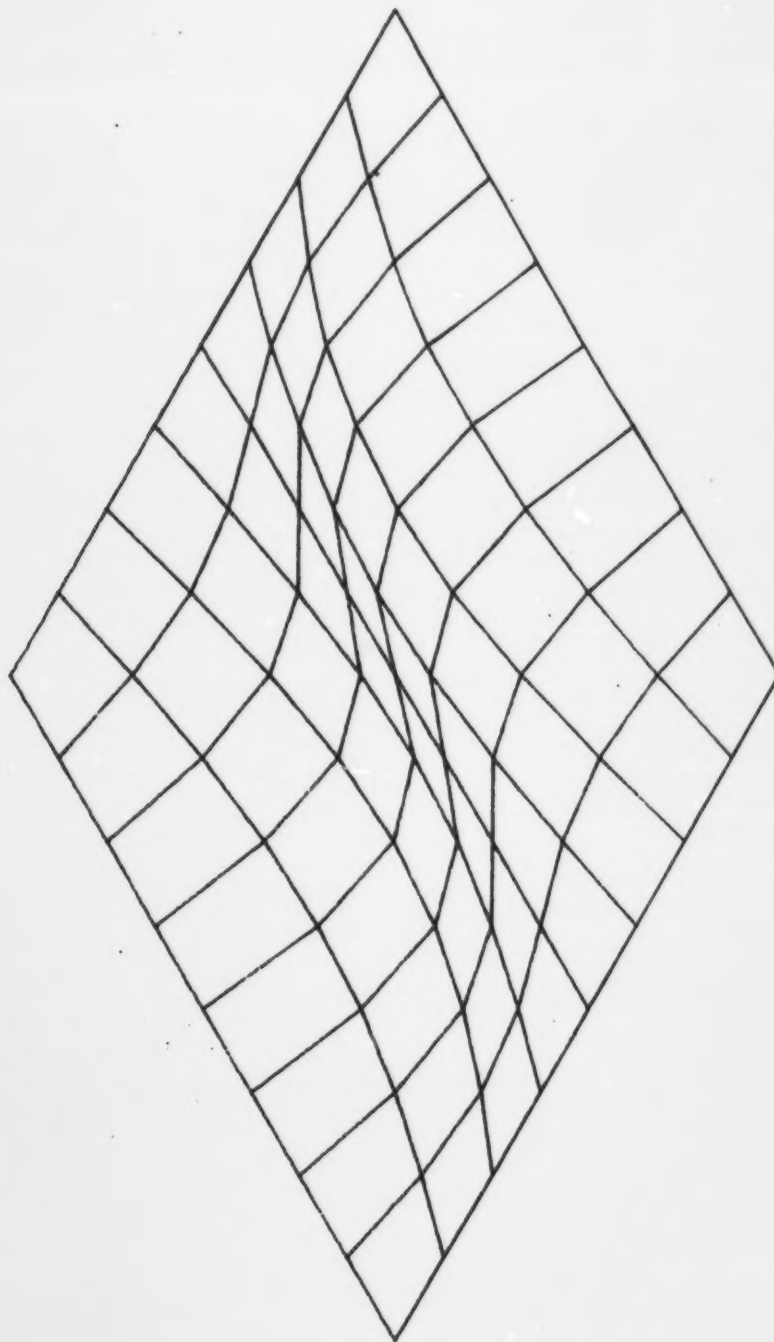
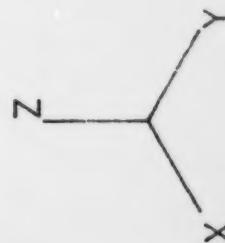


FIGURE 2.25.3

SIMPLY SUPPORTED SQUARE PLATE MODAL ANALYSIS

VIEW POINT 1.0 1.0 1.0 MODE SHAPE 2



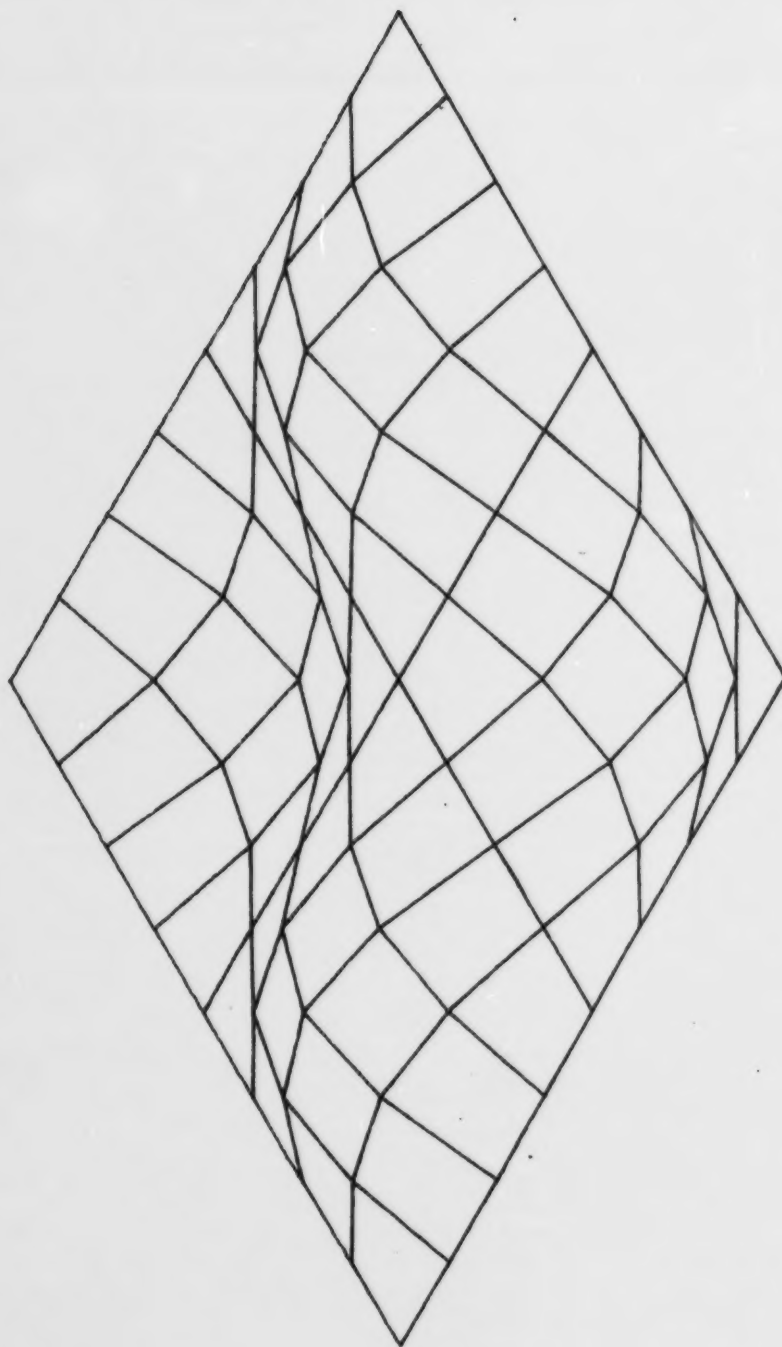
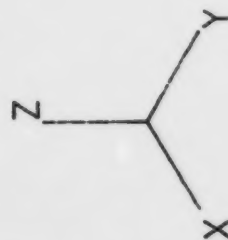


FIGURE 2.25.4

SIMPLY SUPPORTED SQUARE PLATE MODAL ANALYSIS

VIEW POINT 1.0 1.0 1.0 MODE SHAPE 4



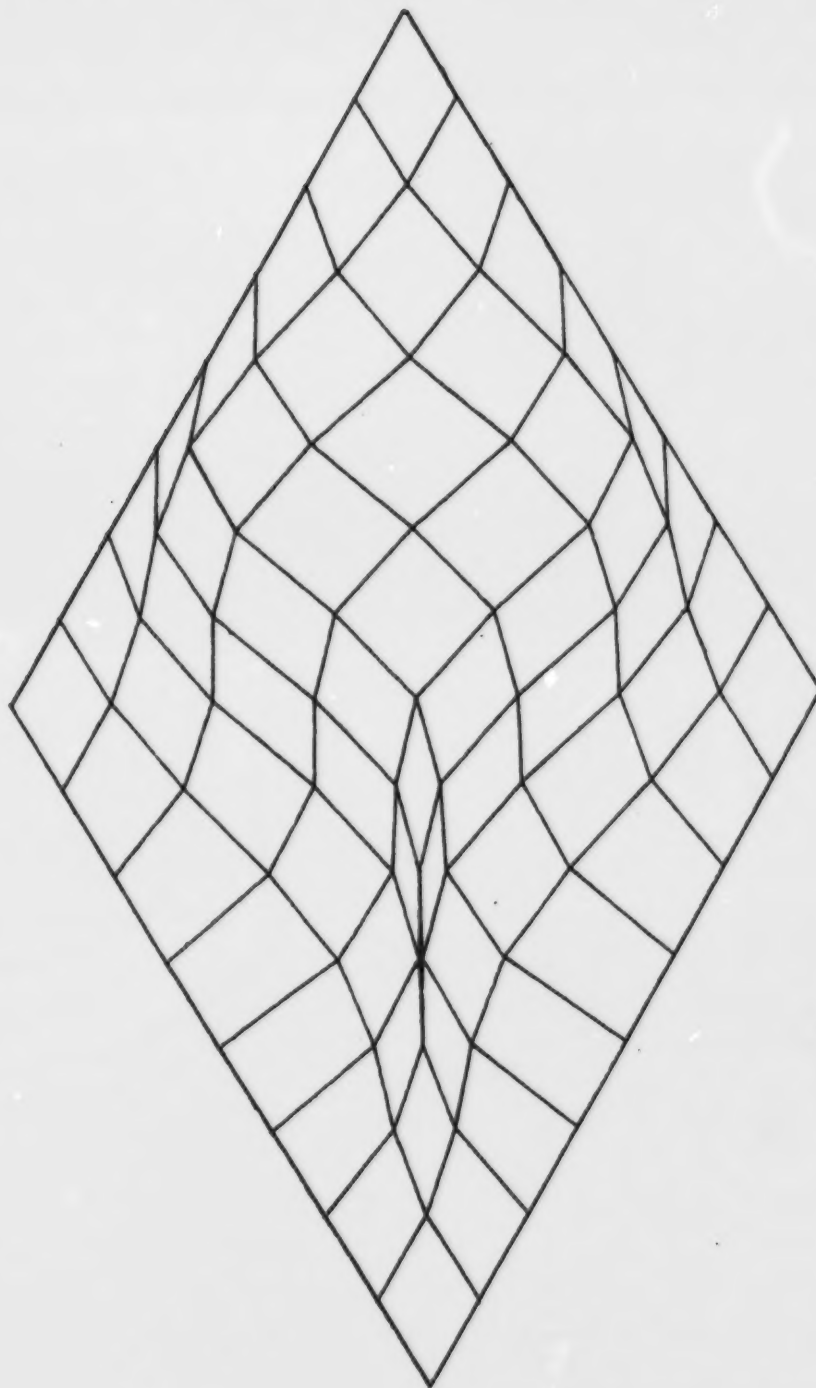
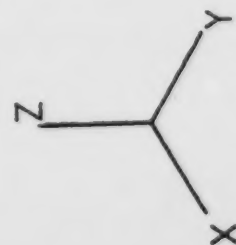


FIGURE 2.25.5
SIMPLY SUPPORTED SQUARE PLATE MODAL ANALYSIS



VIEW POINT 1.0 1.0 1.0 MODE SHAPE 5

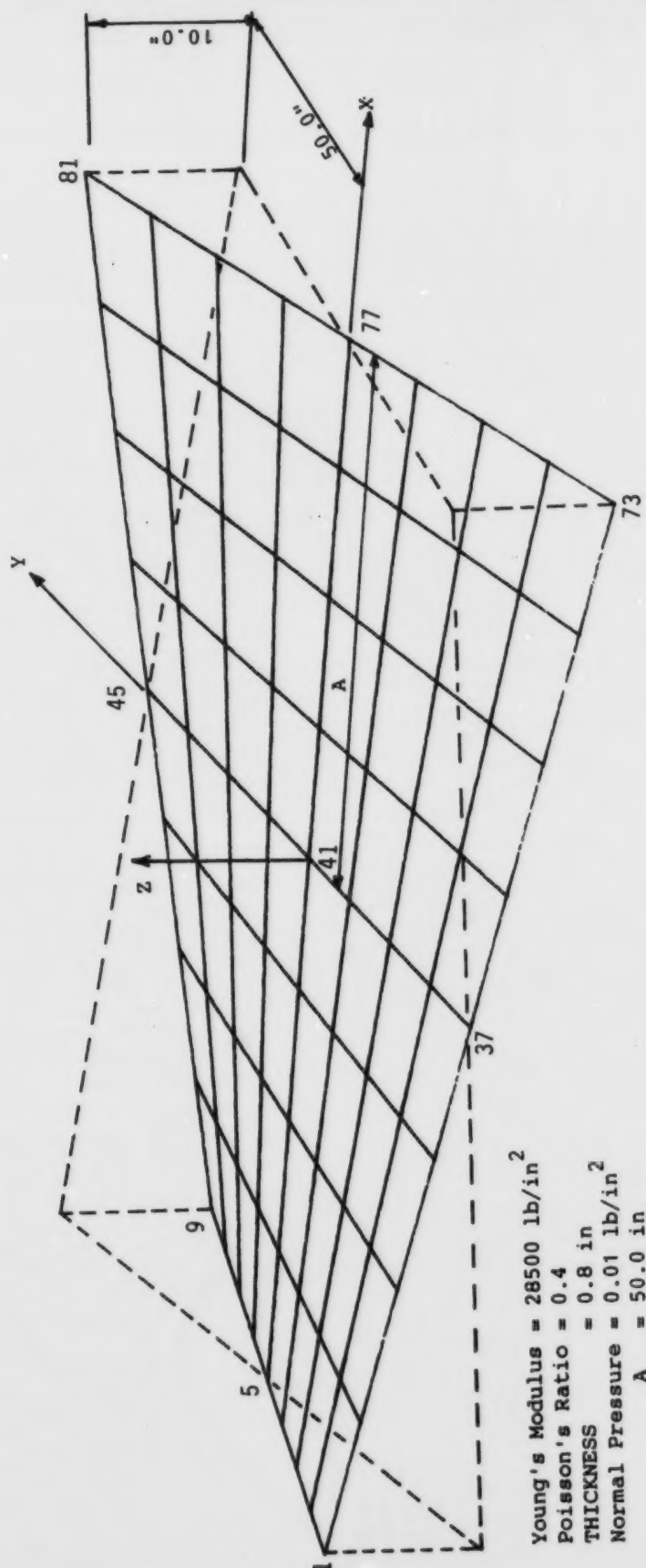
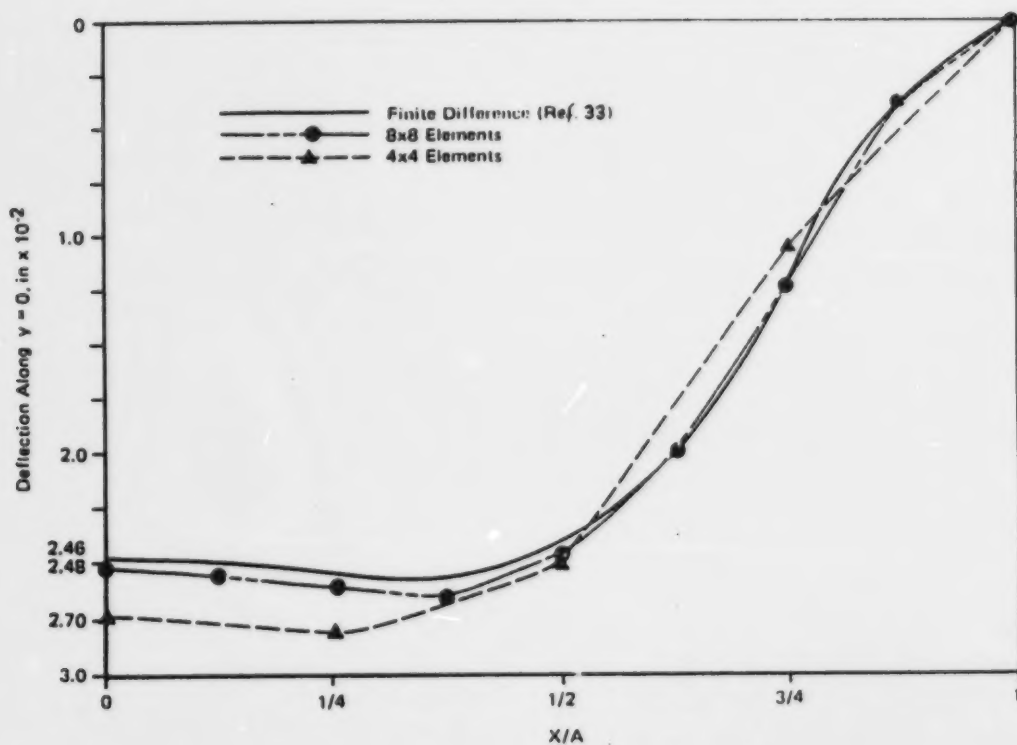
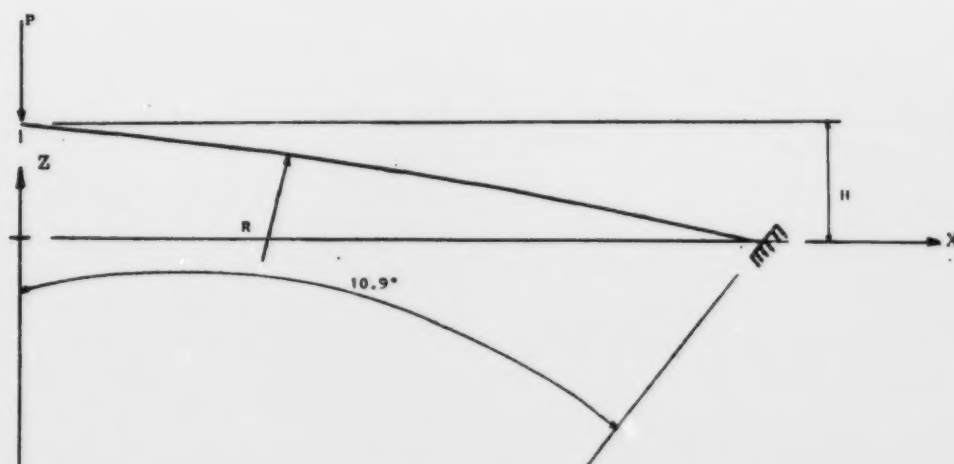
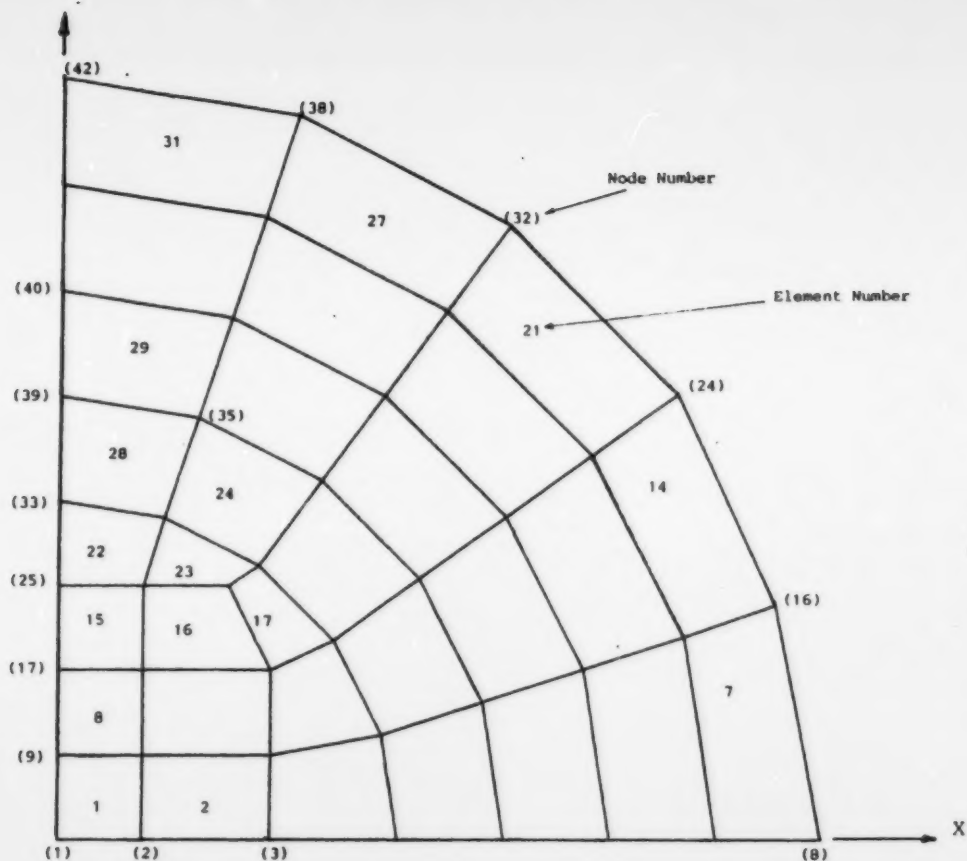


FIGURE 2.26.1
 CLAMPED HYPERBOLIC PARABOLOID SHELL COMPUTER MODEL



X/A	H3DMAP Deflections Along $y=0$, in $\times 10^{-2}$	
	4x4 Mesh	8x8 Mesh
0	2.70	2.48
1/8	..	2.52
1/4	2.79	2.60
3/8	..	2.62
1/2	2.49	2.45
5/8	..	1.99
3/4	1.05	1.23
7/8	..	0.40
1	0.00	0.00

FIGURE 2.26. 2
Deflection at Centre Line of Hyperbolic Paraboloid Shell



Radius = $R = 4.76$ in
 Height = $H = 0.0859$ in
 Young's Modulus = $E = 10 \times 10^6$ lb/in²
 Poisson's Ratio = $\nu = 0.3$
 Density = $\rho = 0.0947$ lb/in³
 Shell Thickness = $t = 0.01576$ in
 Time Span = $\Delta t = 0.0006$ sec
 Initial Time Step = $\Delta t_0 = 2.2 \times 10^{-6}$ sec

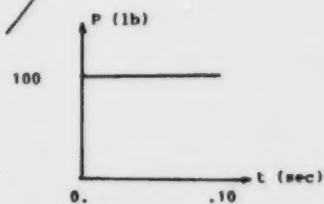


FIGURE 2.27.1

SPHERICAL SHELL DYNAMIC RESPONSE - COMPUTER MODEL

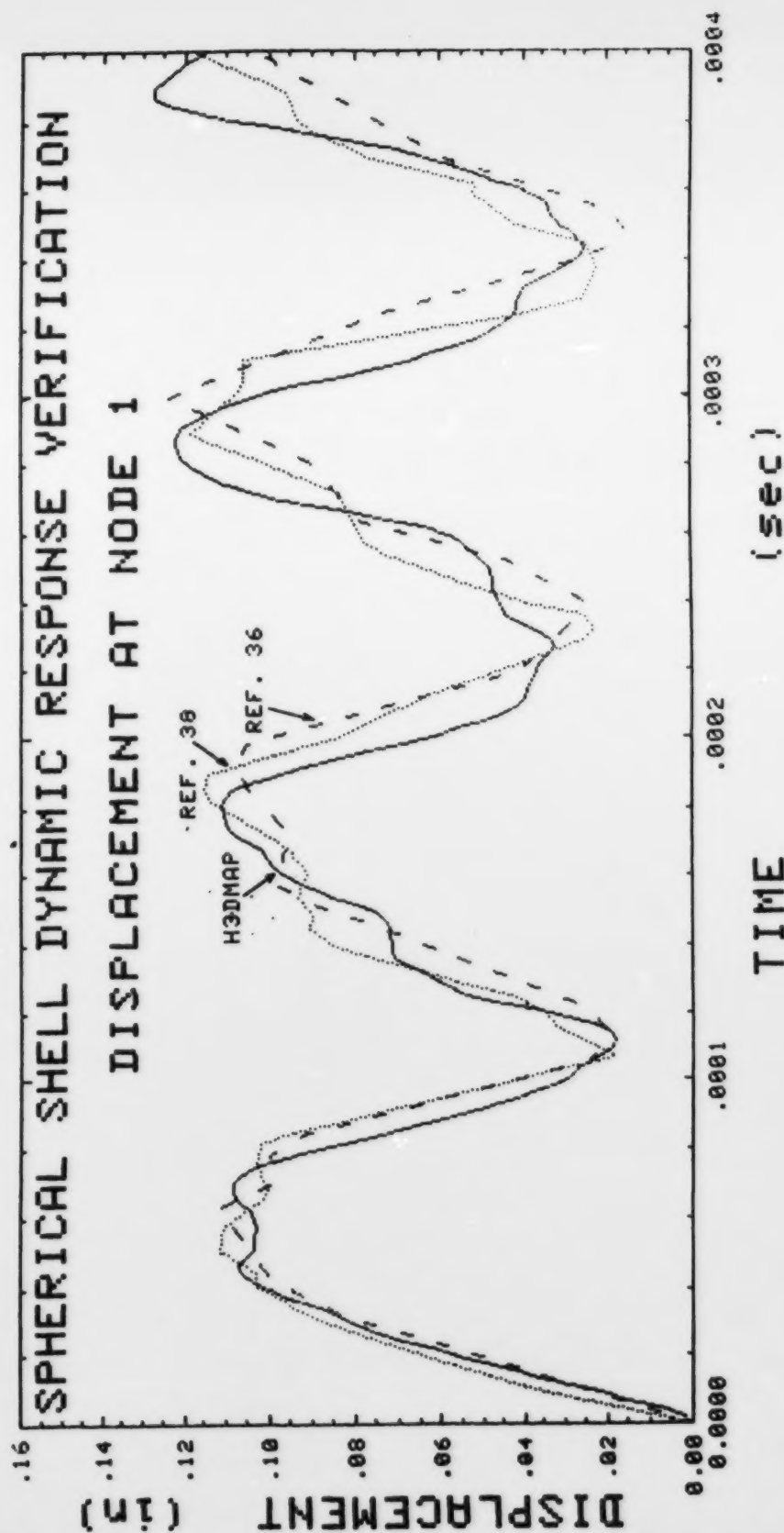


FIGURE 2.27.2

SPHERICAL SHELL DYNAMIC RESPONSE AT NODE 1

	Vertical Displacement (in)				Horizontal Displacement (in)	
	Mesh Size	H3DMAP	Reference 31	Reference 40	H3DMAP	Reference 40
Thin Cylinder	4 x 4 4 x 6	0.02168 0.02313	0.02414*	0.02431	0.020297 0.02135	0.02236
Thick Cylinder	4 x 4 4 x 6	0.1000 0.1076	0.1129			

* 2 x 8 Mesh and 20 d.o.f. per Element

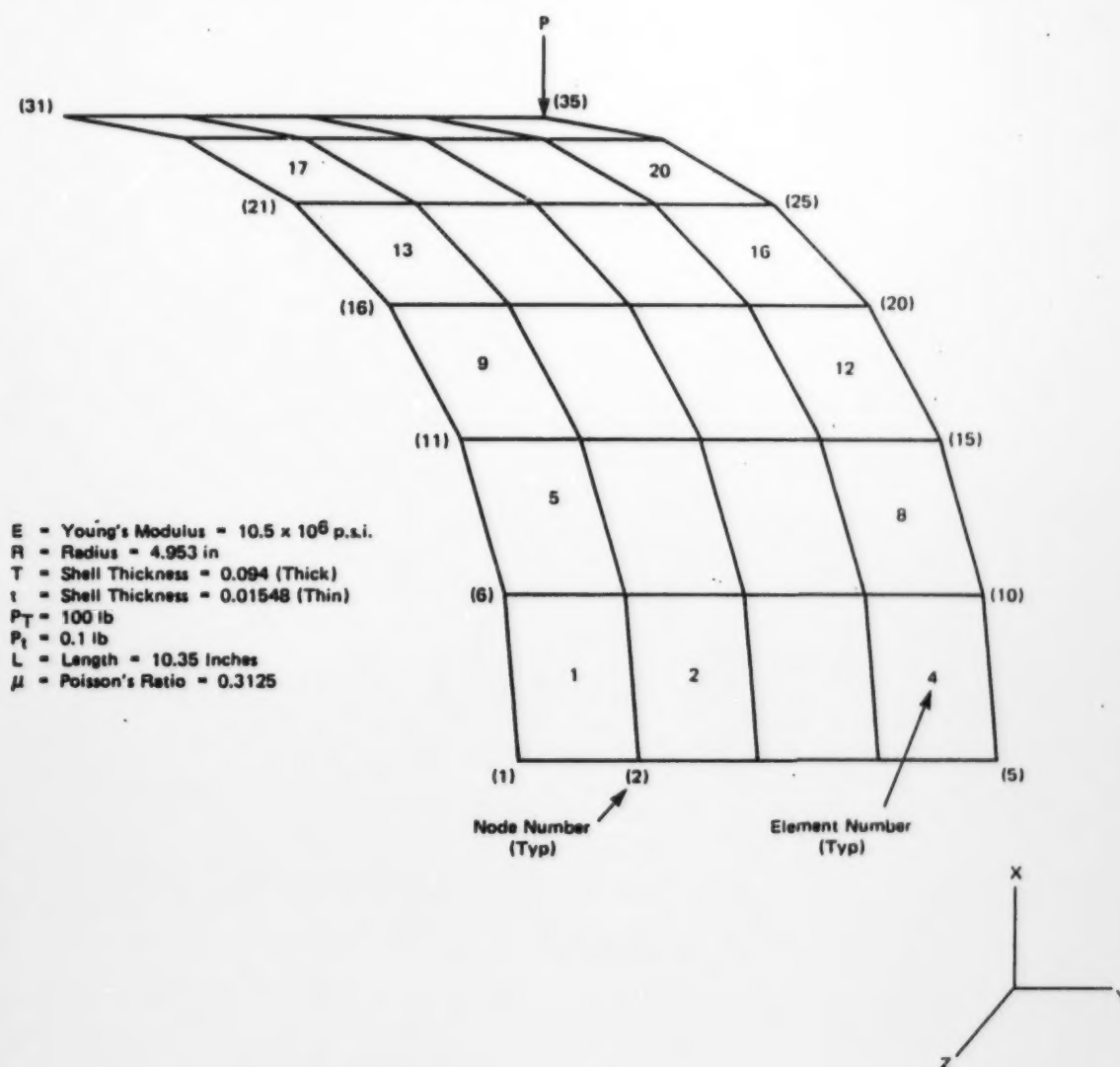
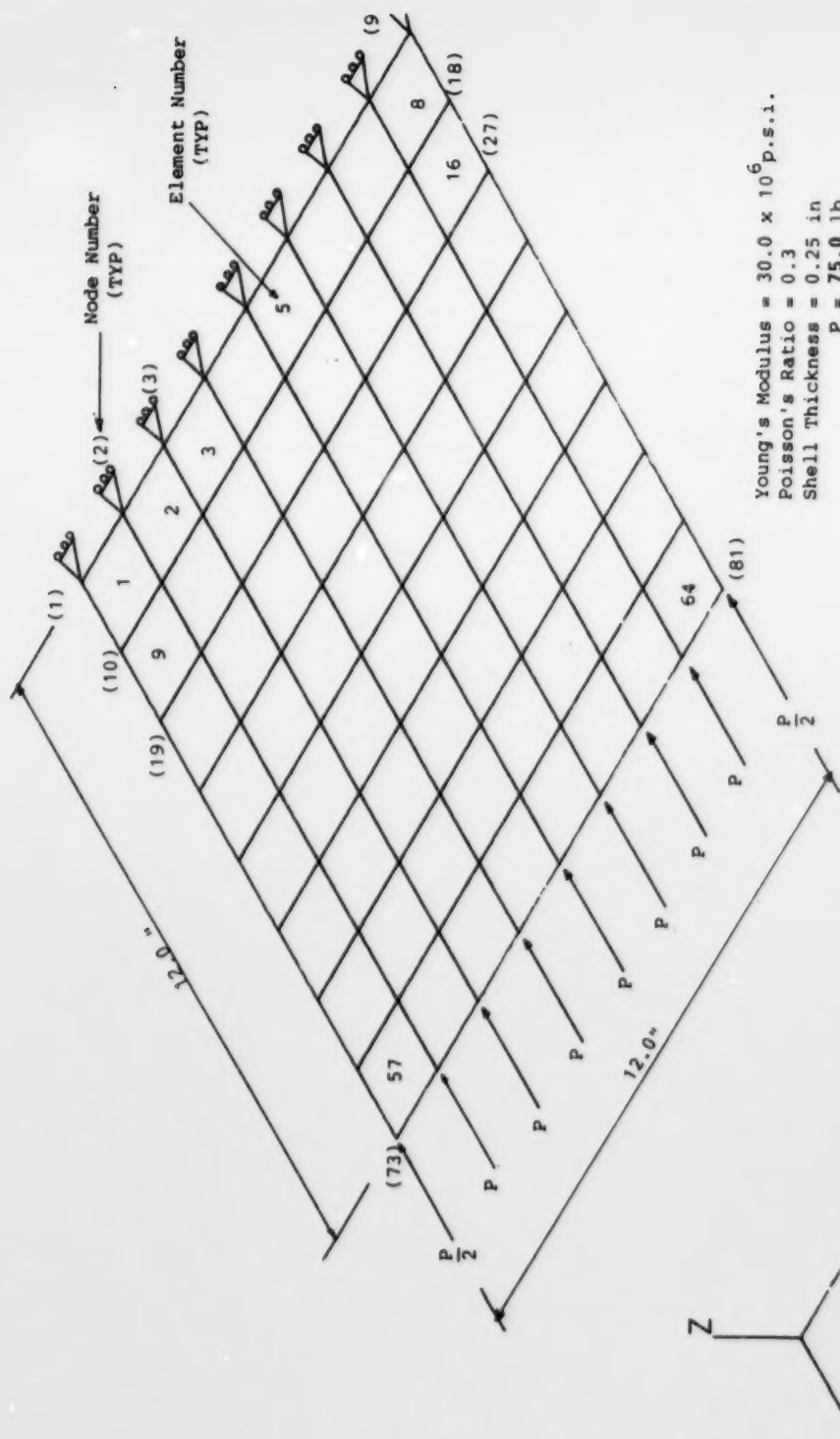


FIGURE 2.28.1
Case 28: Pinched Circular Cylinder - Computer Model



181048-RD

89-221

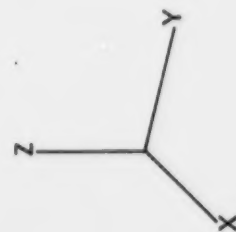
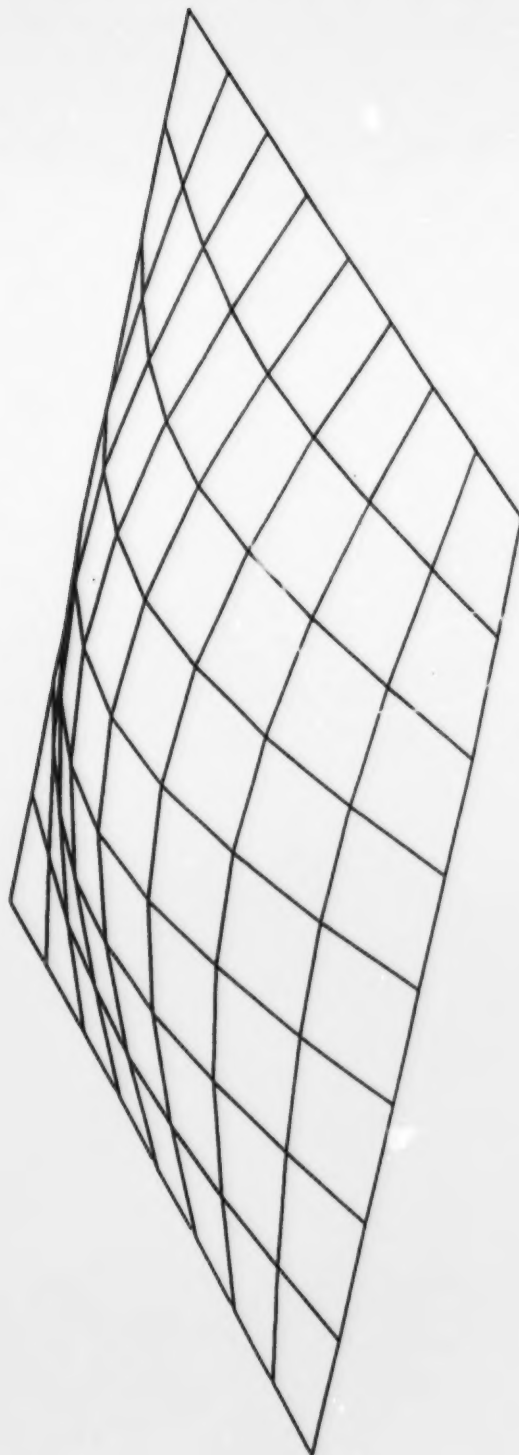
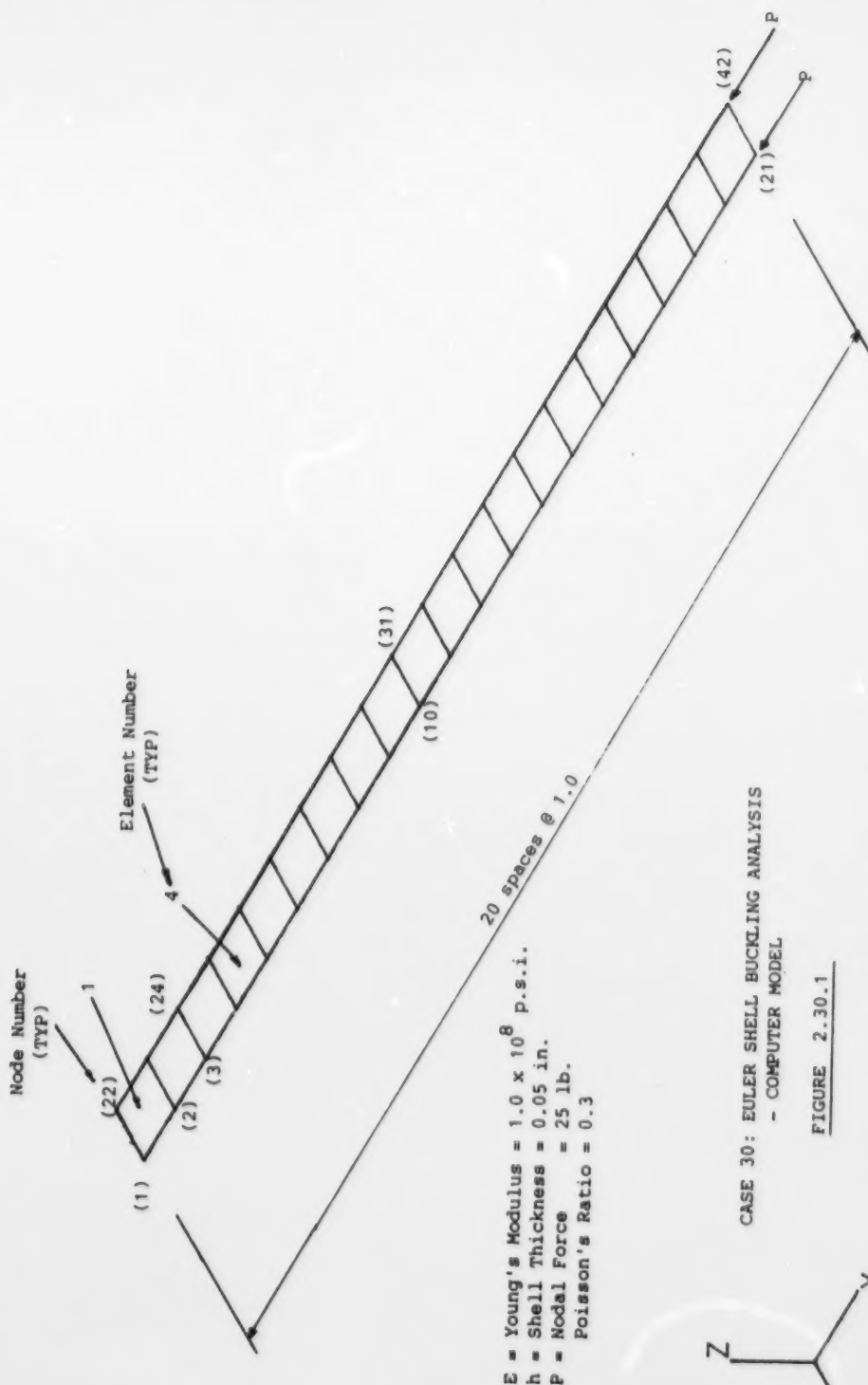


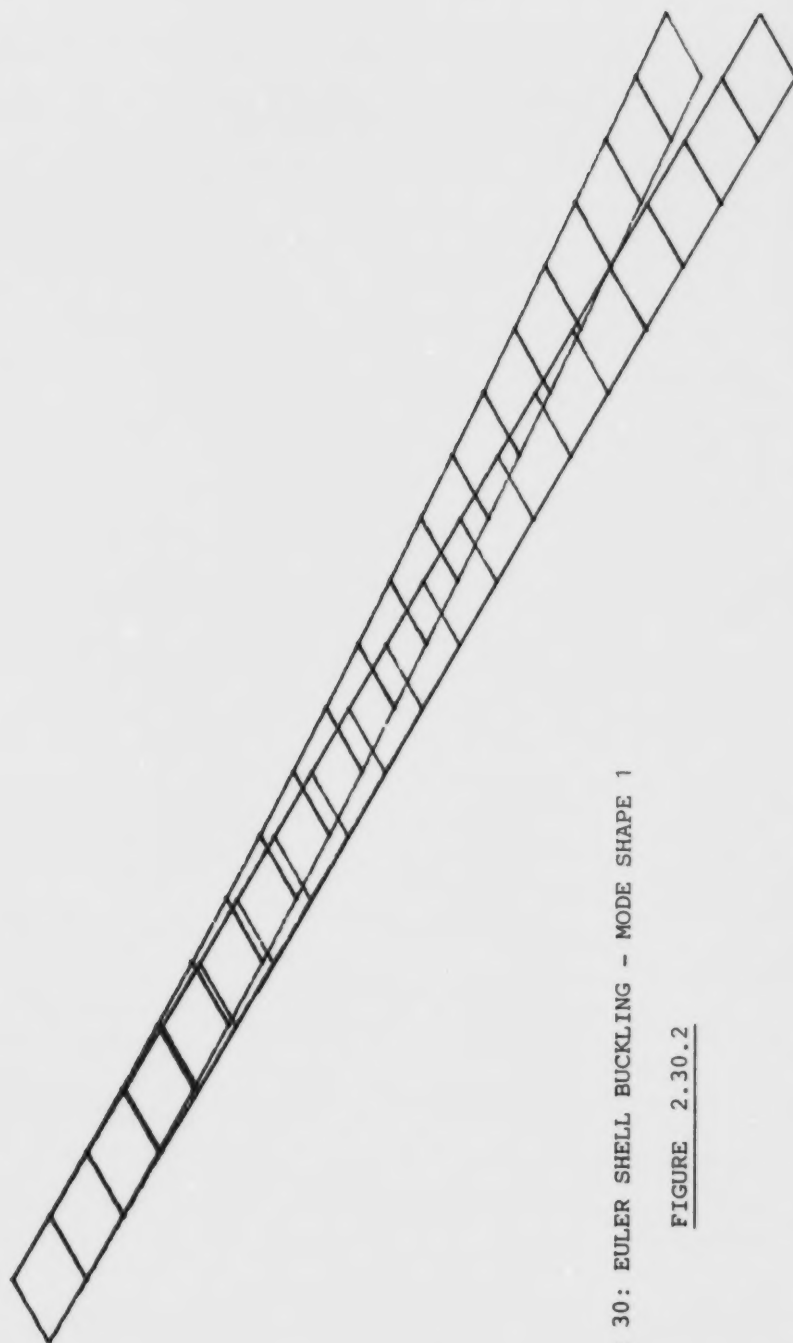
FIGURE 2.29.2
SQUARE PLATE STABILITY ANALYSIS - MODE SHAPE 1



E = Young's Modulus = 1.0×10^8 p.s.i.
h = Shell Thickness = 0.05 in.
P = Nodal Force = 25 lb.
Poisson's Ratio = 0.3

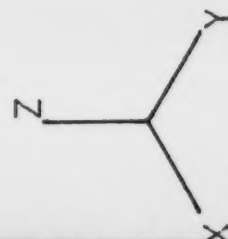
CASE 30: EULER SHELL BUCKLING ANALYSIS
- COMPUTER MODEL

FIGURE 2.30.1

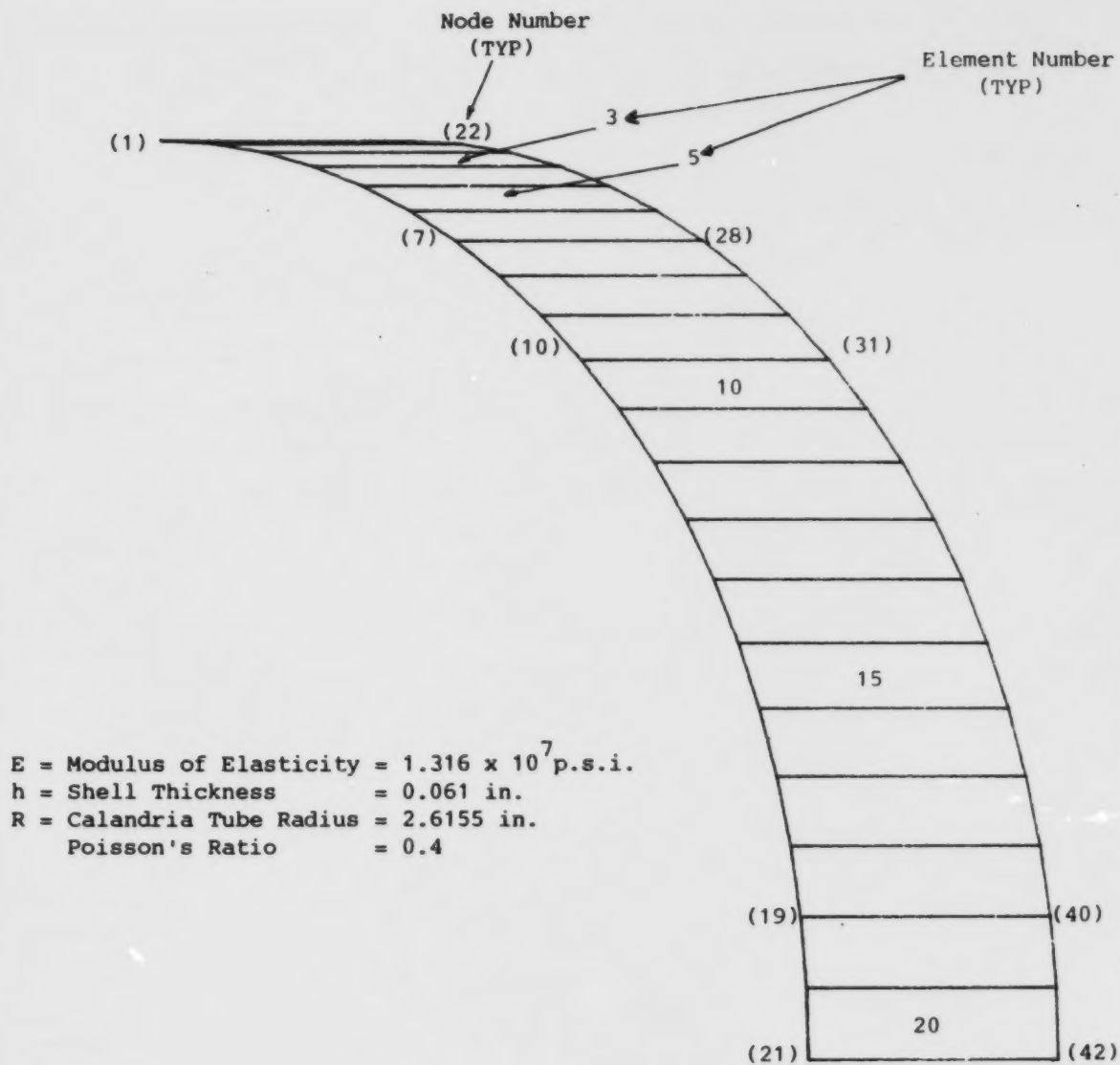


CASE 30: EULER SHELL BUCKLING - MODE SHAPE 1

FIGURE 2.30.2



VIEW POINT 1.0 1.0 1.0 MODE SHAPE 1



VIEW POINT 1.0 0.0 1.0
SHELL VERIF. CASE - C/T QUART MODEL (20 ELEM)

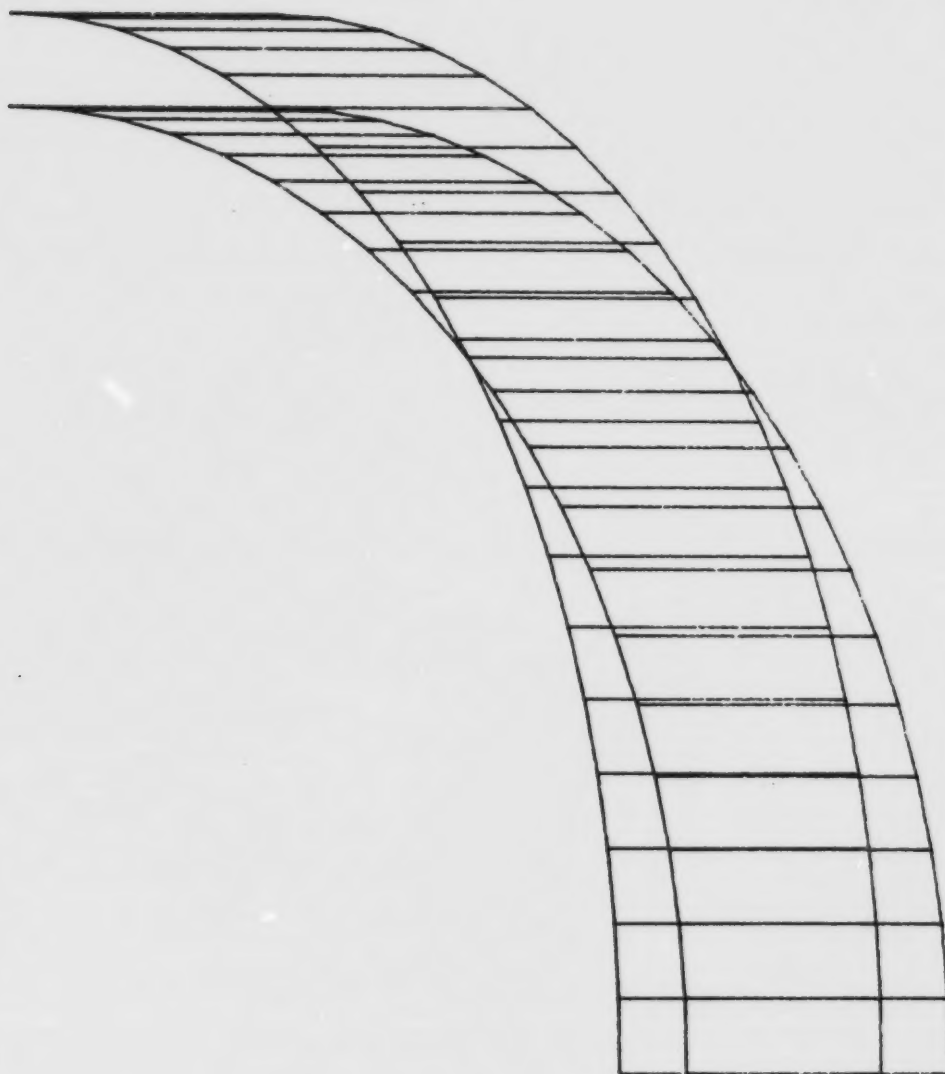


CASE 31: CALANDRIA TUBE SHELL BUCKLING ANALYSIS -
COMPUTER MODEL (1/4 MODEL)

FIGURE: 2.31.1

CASE 31: CALANDRIA TUBE SHELL BUCKLING ANALYSIS
- MODE SHAPE 1

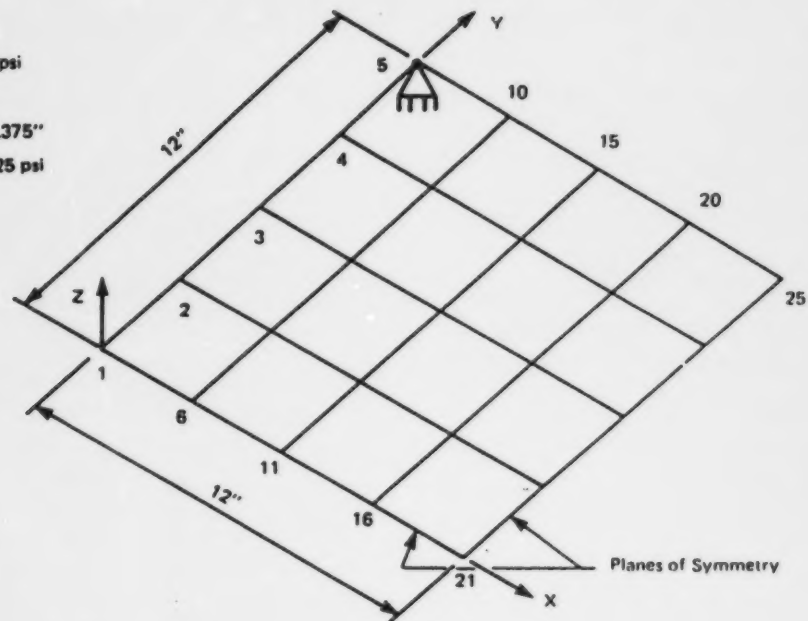
FIGURE 2.31.2



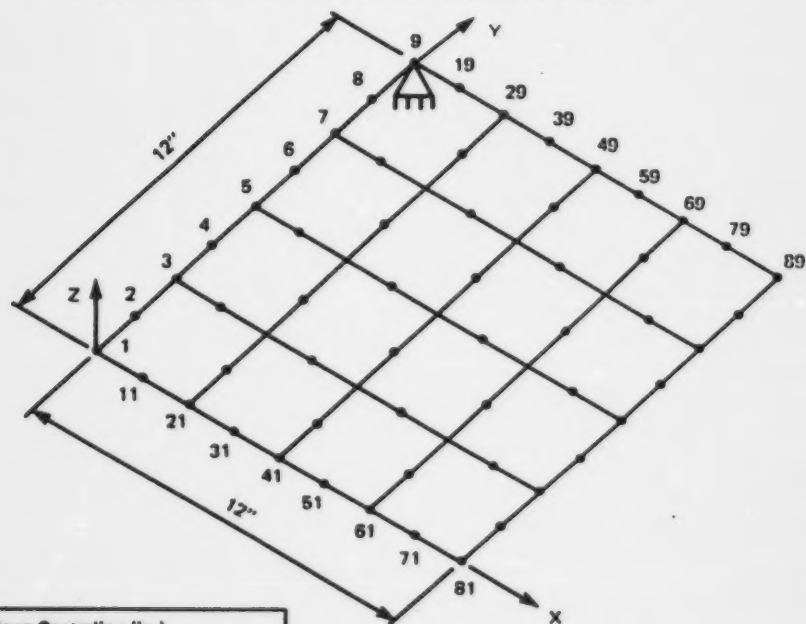
VIEW POINT 1.0 0.0 1.0 MODE SHAPE 1
SHELL VERIF. CASE - C/T QUART MODEL (20 ELEM)

H3DMAP 4-NODED SHELL ELEMENT (1/4 MODEL)

$E = 4.3 \times 10^6$ psi
 $\mu = 0.38$
 Thickness = 0.375"
 UDL = -0.3125 psi

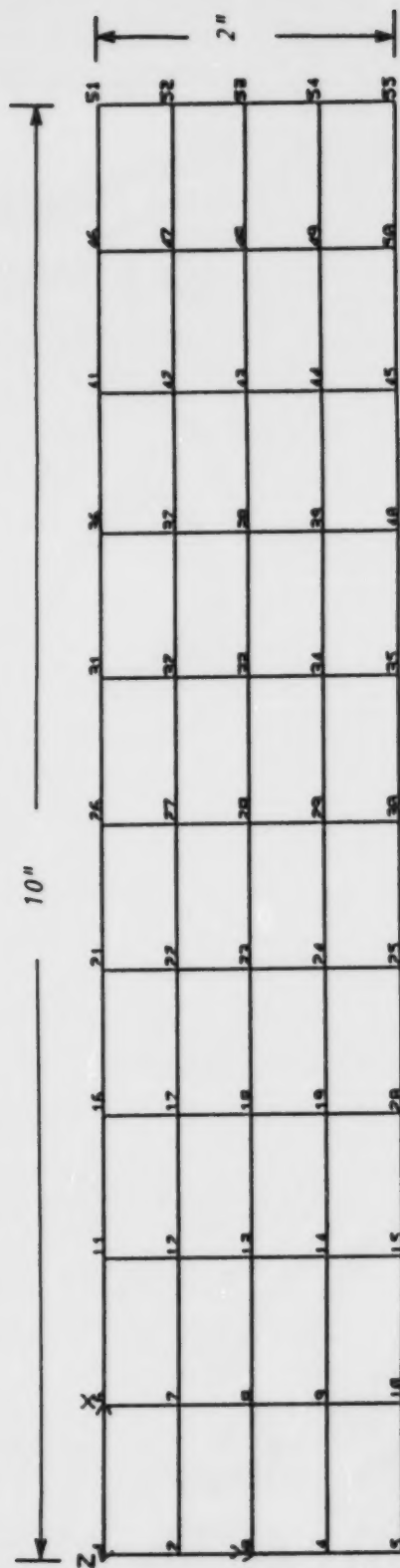


ABAQUS 8-NODED QUADRATIC SHELL ELEMENT (1/4 MODEL)

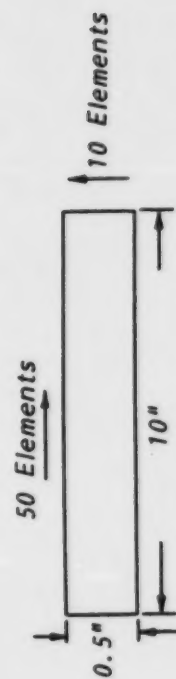


Z-Deflection Along Centreline (in.)				
Node	H3DMAP	Node	ABAQUS	Classical Theory
1	-0.086301	1	-0.08849	-0.12065
6	-0.10489	21	-0.09908	
11	-0.10611	41	-0.1102	
16	-0.12274	61	-0.1183	
21	-0.11648	81	-0.1212	

FIGURE 2.32.1
 Case 32: Corner Supported Plate Test



(a)



(b)

FIGURE 2.34.1
CASE 34: 2-D CANTILEVER BEAM MODELS

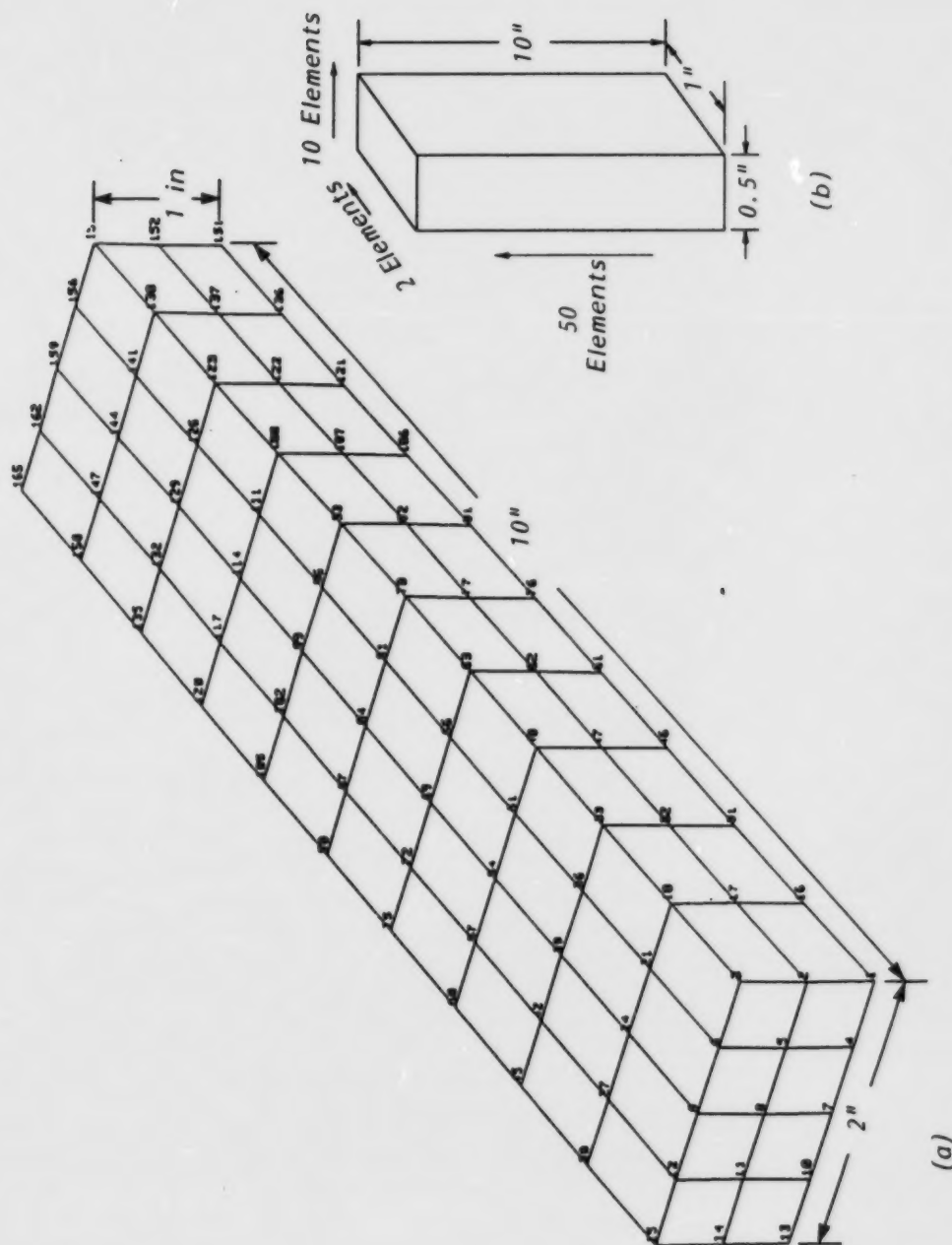


FIGURE 2.34.2
CASE 34: 3-D CANTILEVER BEAM MODELS

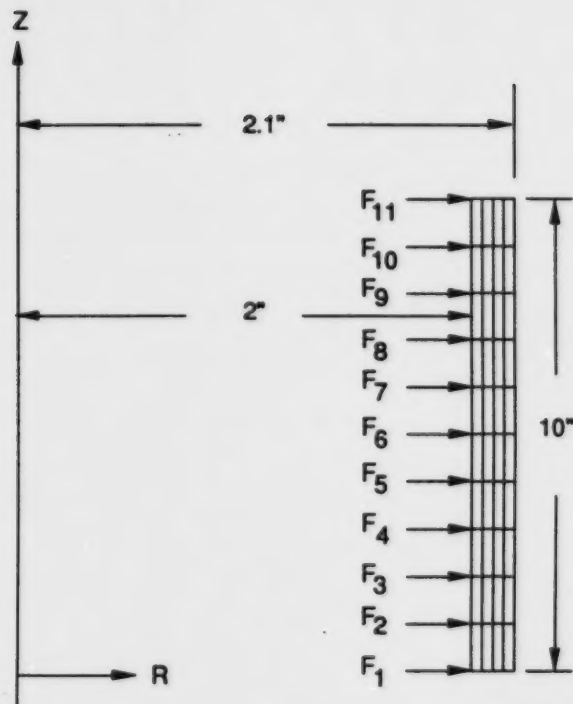


FIGURE 2.35.1

CASE 35: THIN WALL VESSEL MODEL

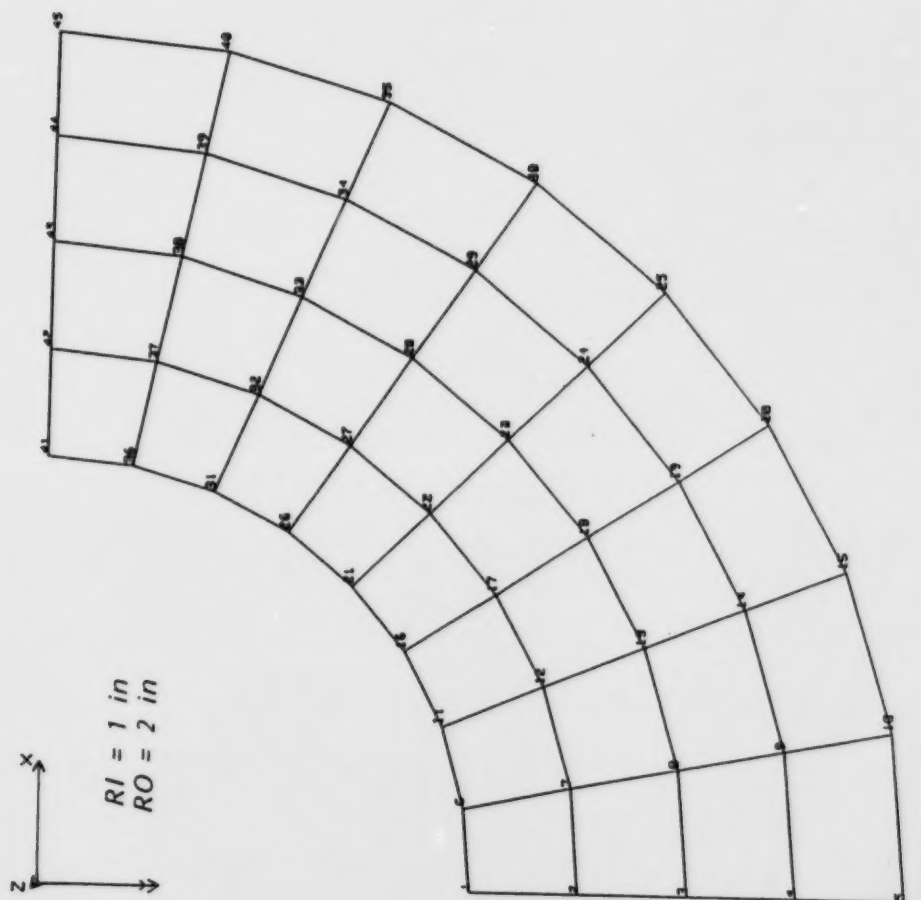


FIGURE 2.36.1
CASE 36: THICK WALL CYLINDER MODEL

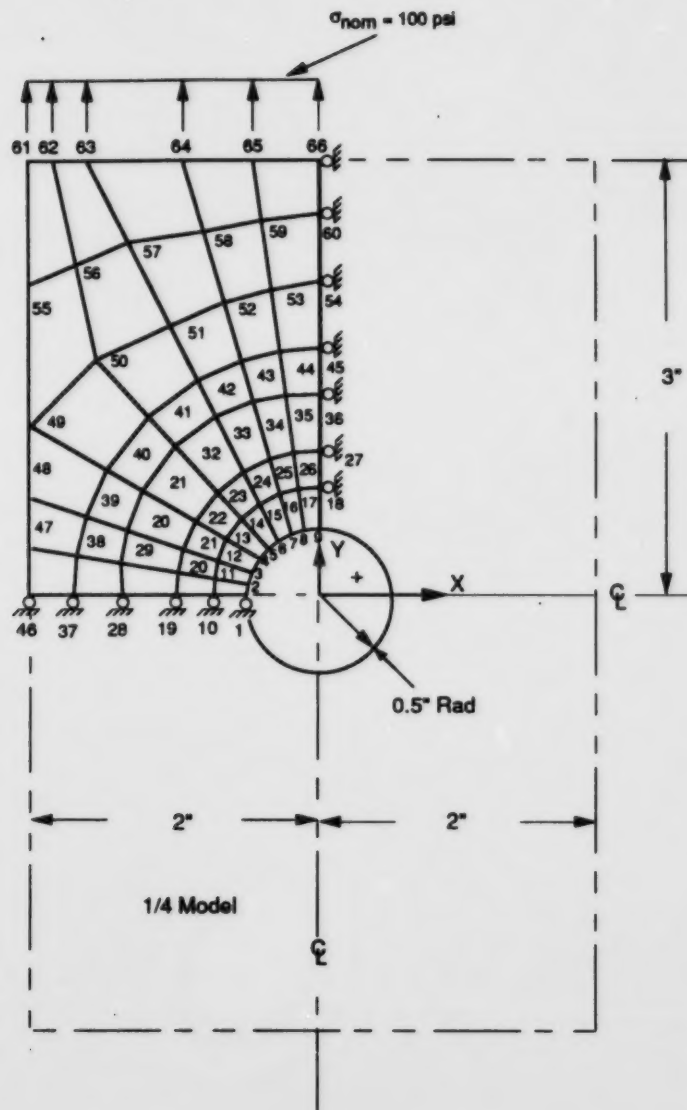


FIGURE 2.37.1

CASE 37: STRESS CONCENTRATION EVALUATION
COMPUTER MODEL



1st Mode Shape

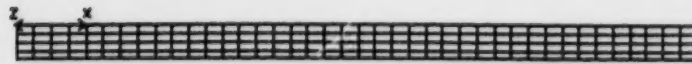
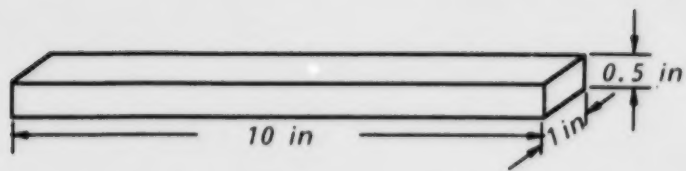


2nd Mode Shape



3rd Mode Shape

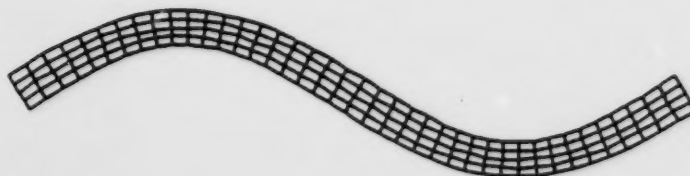
FIGURE 2.38.1
CASE 38: SIMPLY SUPPORTED CIRCULAR DISK
FIRST THREE MODE SHAPES



Undeformed Geometry



1st Mode Shape



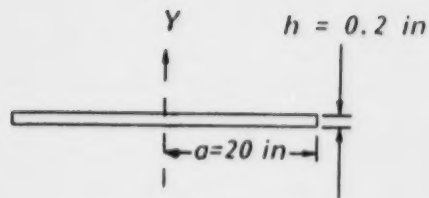
2nd Mode Shape



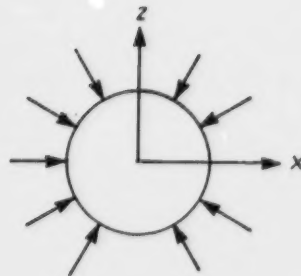
3rd Mode Shape

FIGURE 2.39.1
CASE 39: SIMPLY SUPPORTED BEAM
FIRST THREE MODE SHAPES

181541-RD



Geometry



Applied Load



Computer Model



1st Mode Shape

FIGURE 2.40.1
CASE 40: AXISYMMETRIC BUCKLING OF A CIRCULAR PLATE
GEOMETRY, MODEL, LOAD & 1st MODE SHAPE

Material: 6061-T6 Aluminum

$$E = 10.4 \times 10^6 \text{ psi}$$

$$\nu = 0.3$$

$$\rho = 0.101 \text{ lb/in}^3$$

$$\sigma_y = 41400 \text{ lb/in}^2$$

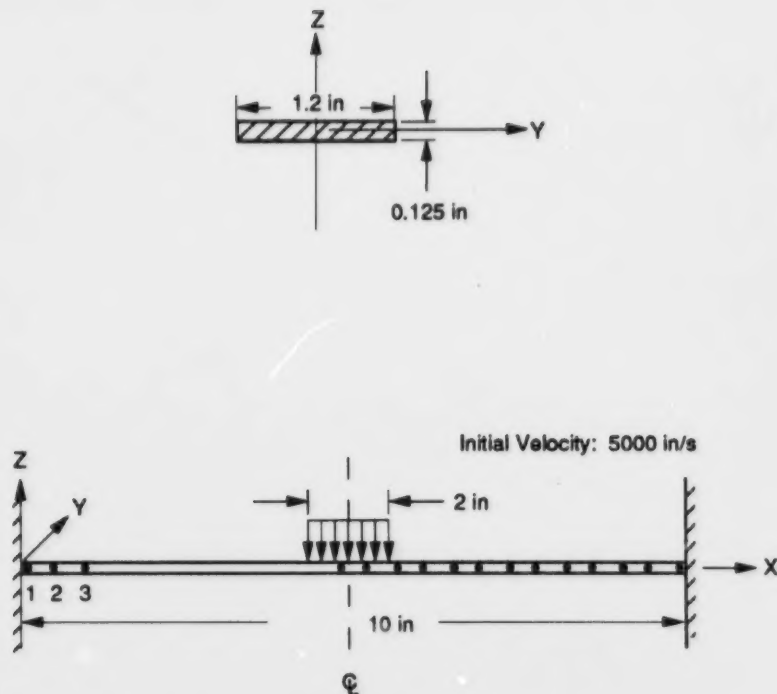


FIGURE 2.41.1

CASE 41: IMPULSIVELY LOADED BEAM MODEL

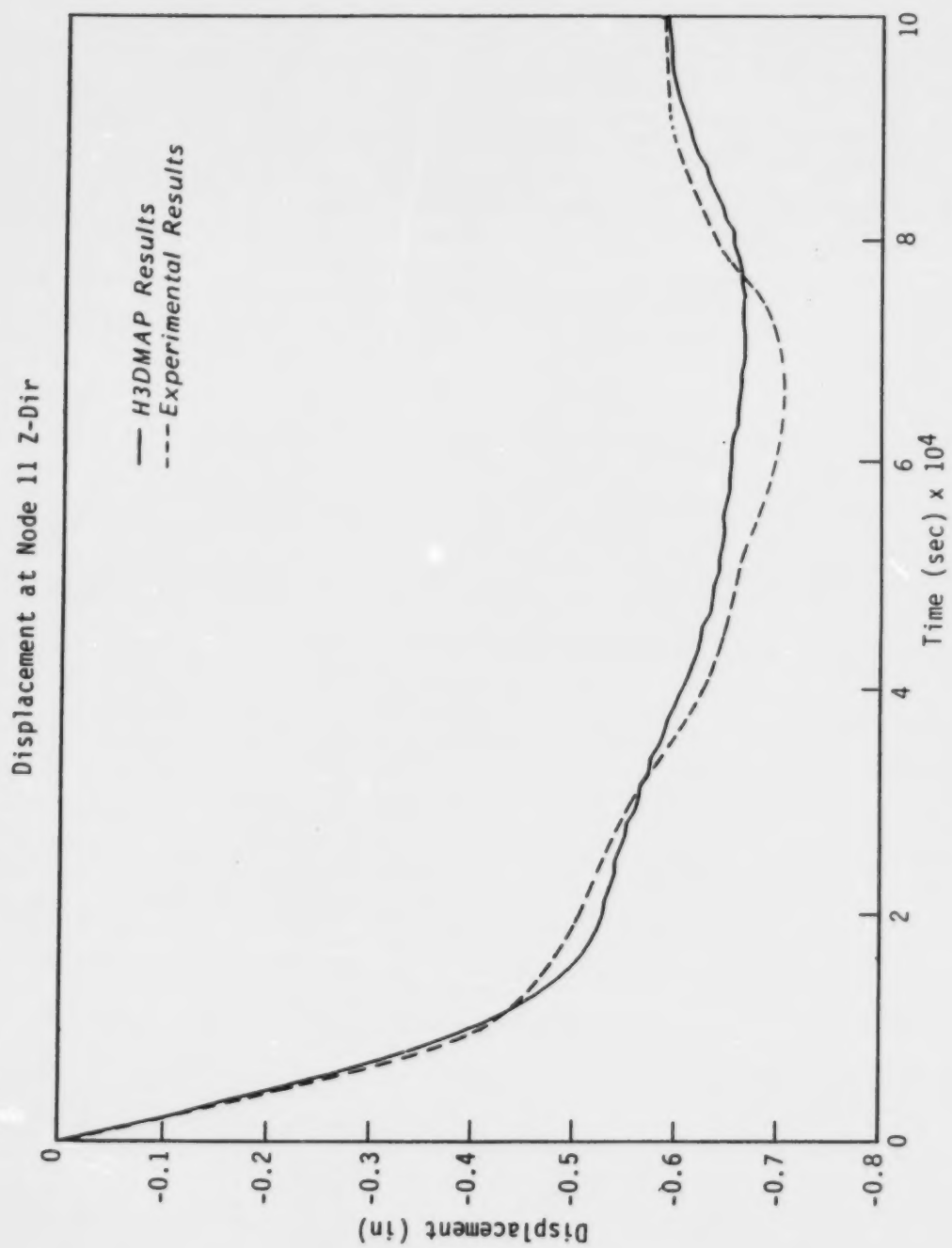


FIGURE 2.41.2
CASE 41: IMPULSIVELY LOADED BEAM

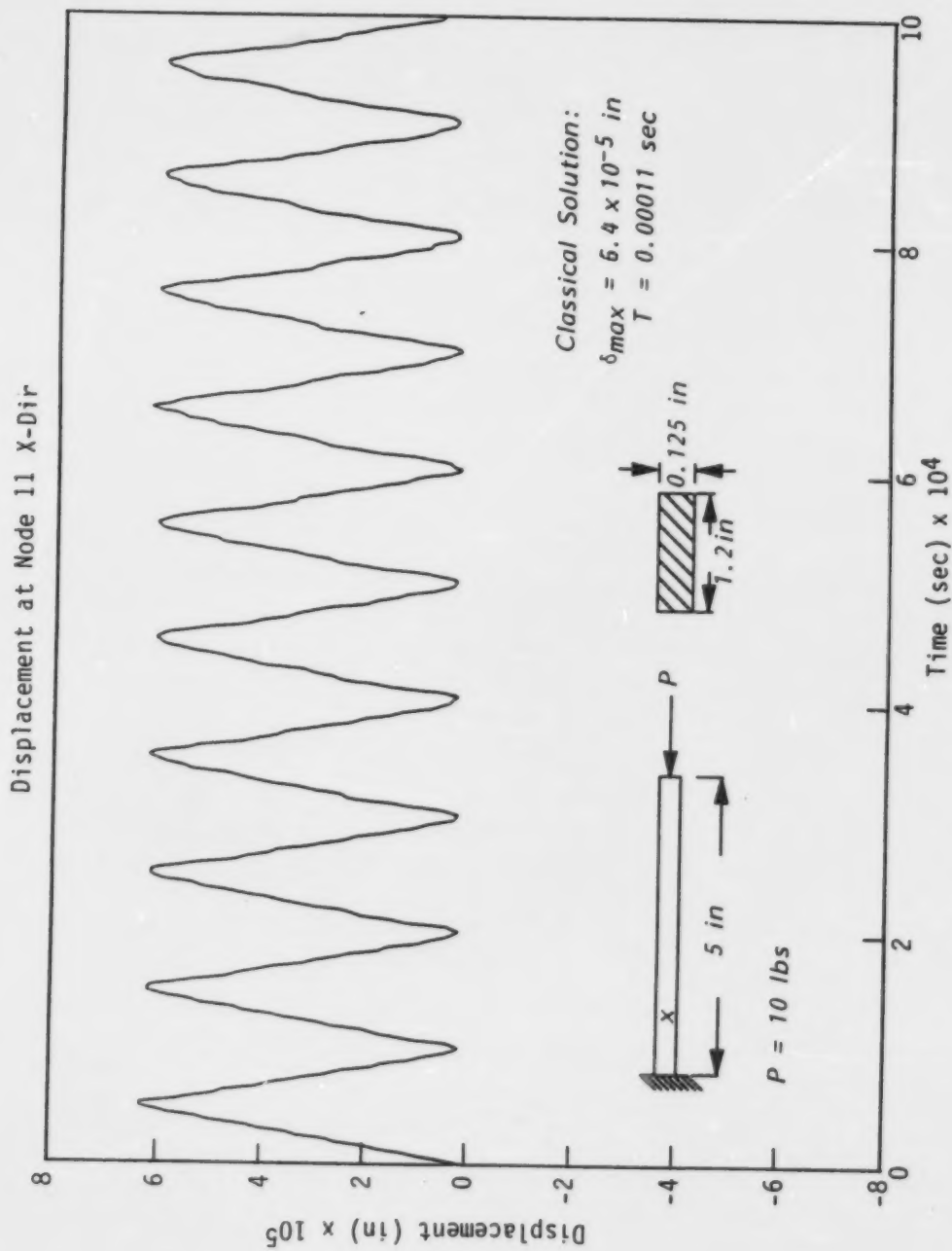


FIGURE 2.42.1
 CASE 42: IMPULSIVELY AXIALLY LOADED BEAM

Geometric Non-Linear Response--Tip Defl. at Node 6 Y-Dir

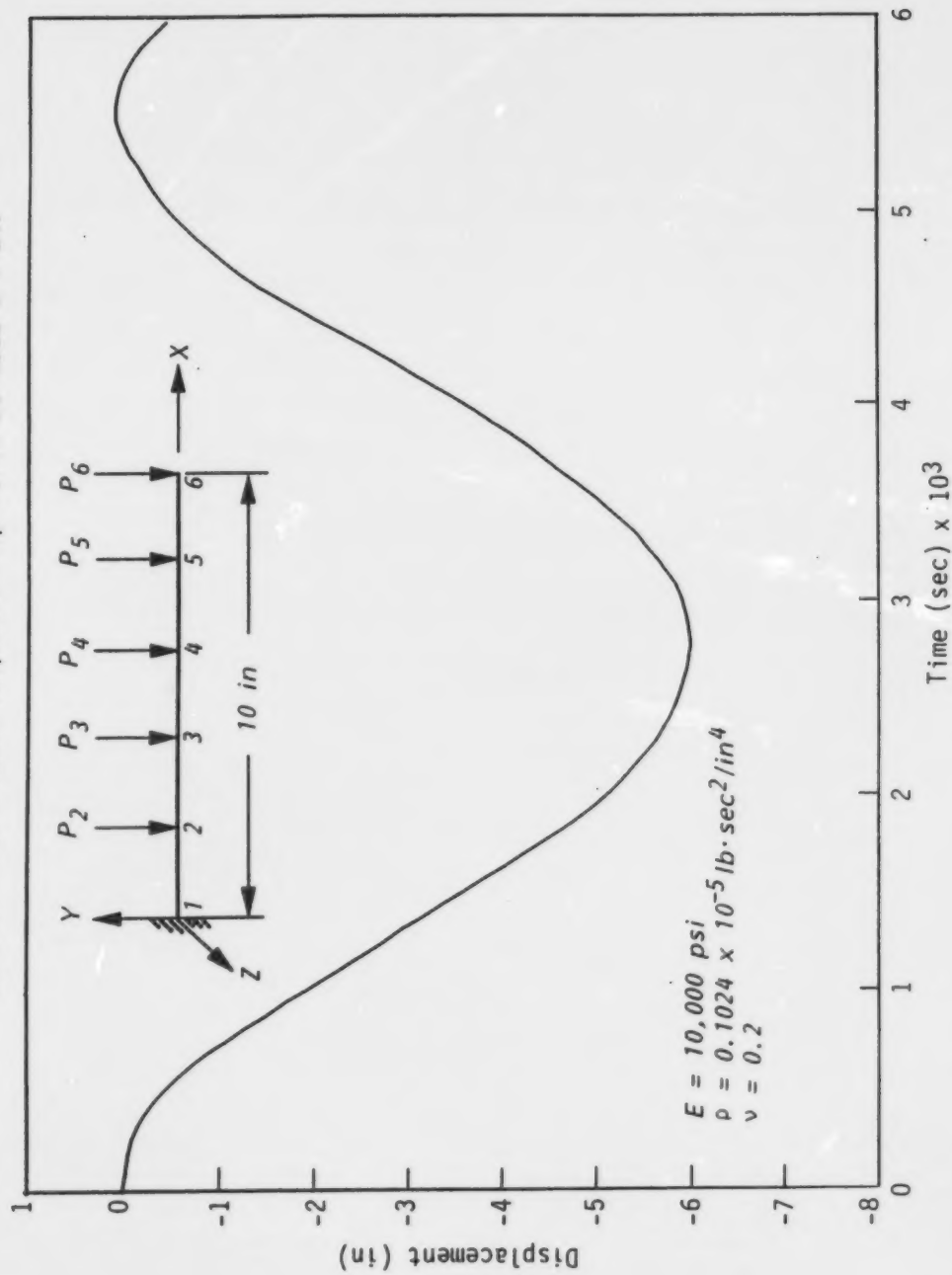


FIGURE 2.43.1
CASE 43: GEOMETRIC NON-LINEAR BEAM TEST



FIGURE 2.43.2
CASE 43: GEOMETRIC NON-LINEAR BEAM TEST
RESPONSE AT TIME: 0.0027 sec

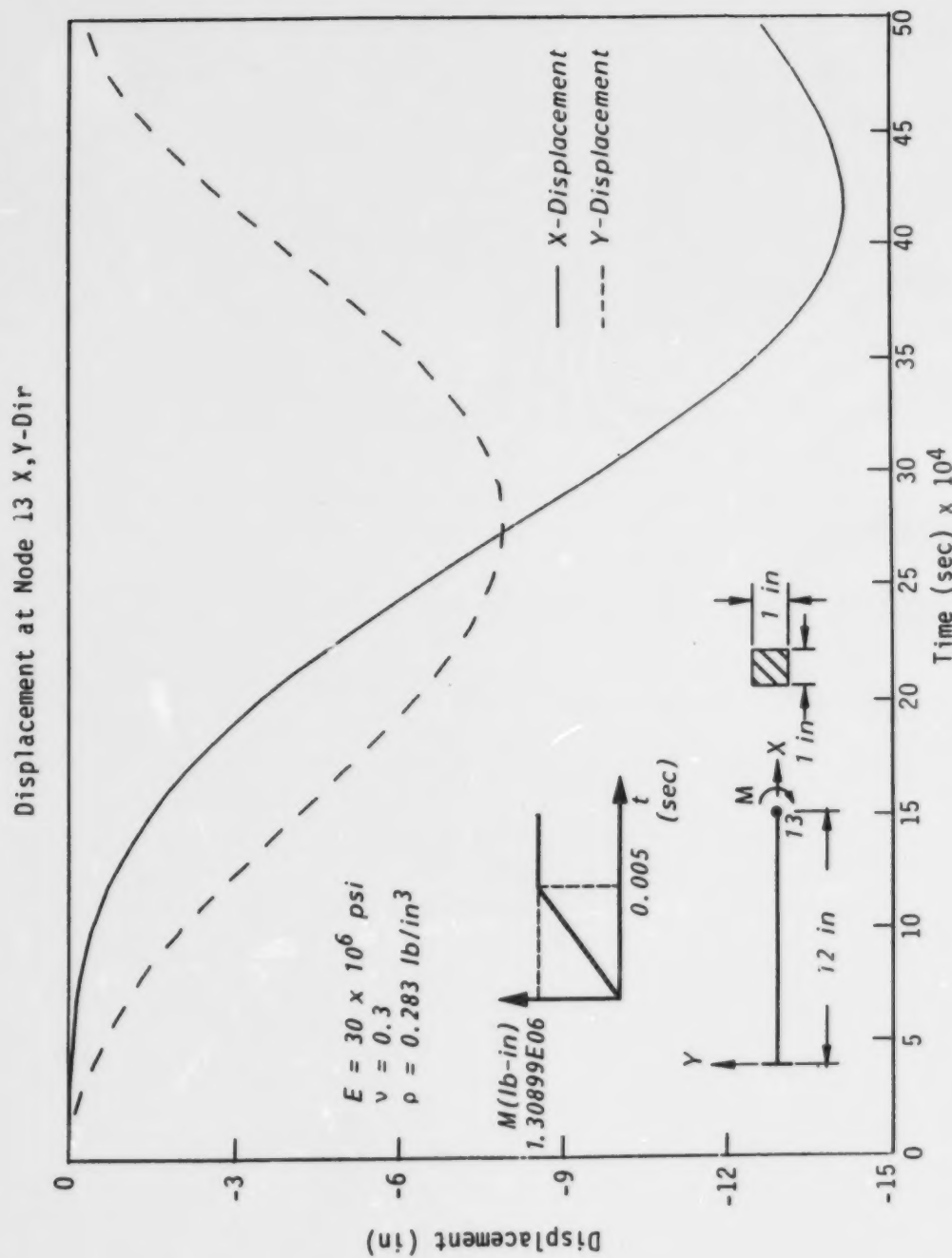


FIGURE 2.44.1
CASE 44: MOMENT LOAD TEST FOR NON-LINEAR BEAM

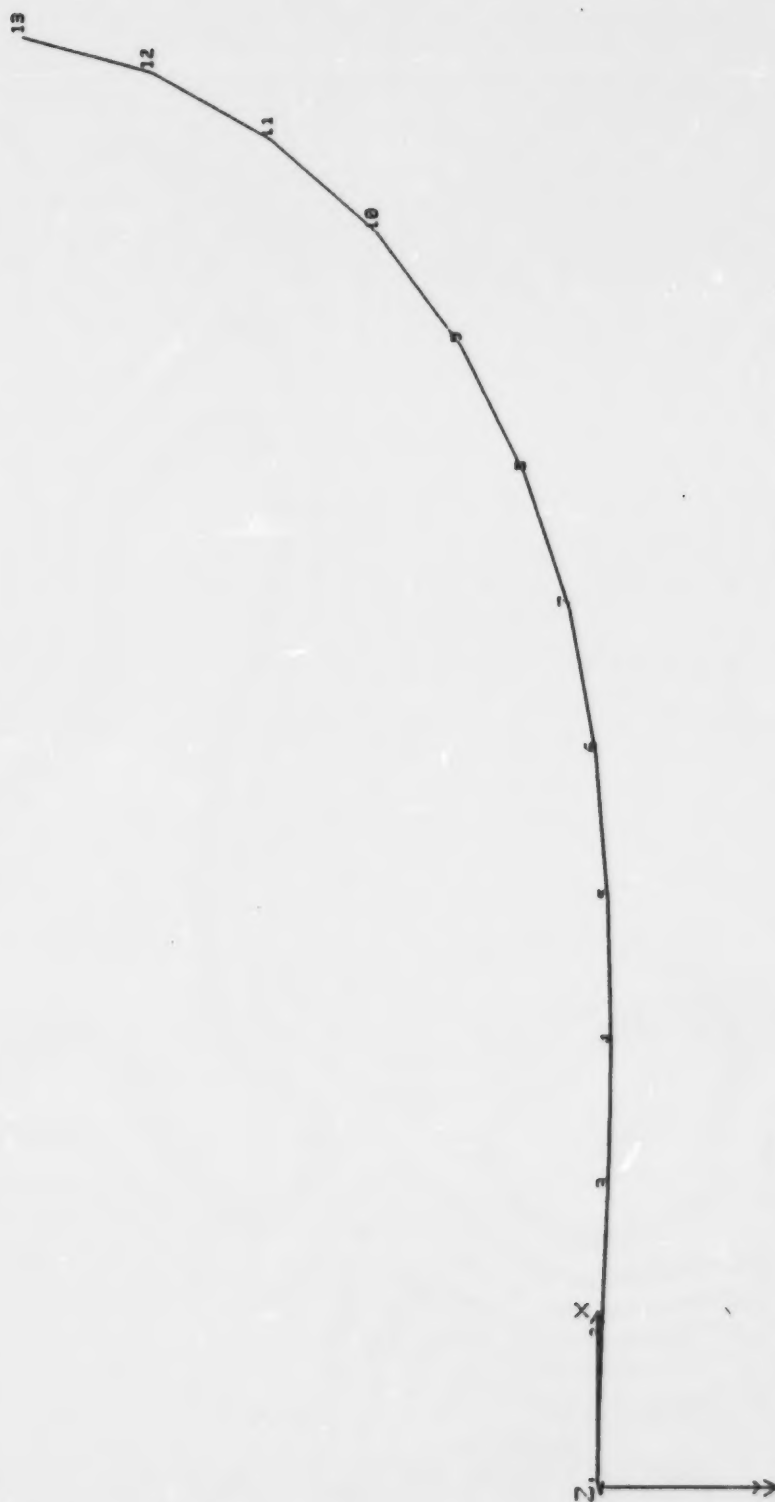


FIGURE 2.44.2
CASE 44: MOMENT LOAD TEST FOR NON-LINEAR BEAM
RESPONSE AT TIME: 0.0010 sec

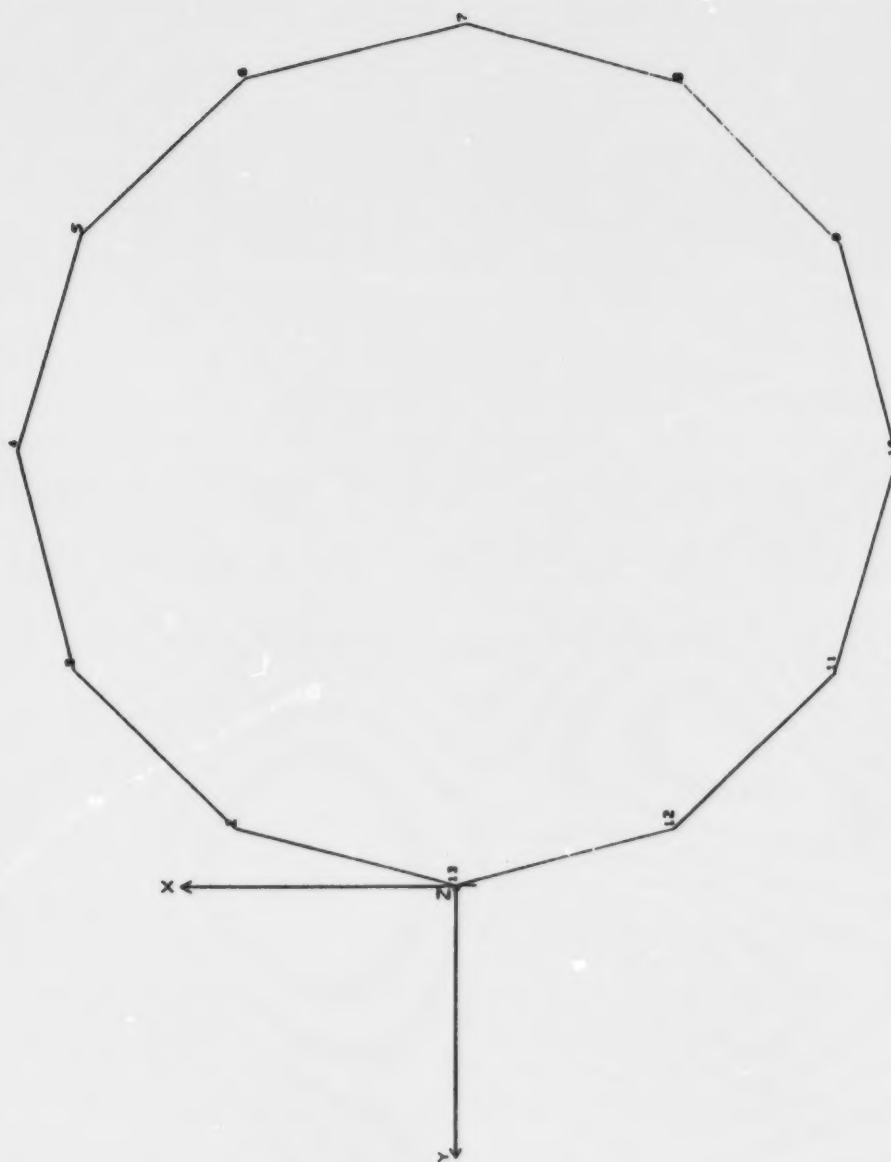


FIGURE 2.44.3
CASE 44: MOMENT LOAD TEST FOR NON-LINEAR BEAM
RESPONSE AT TIME: 0.0050 sec

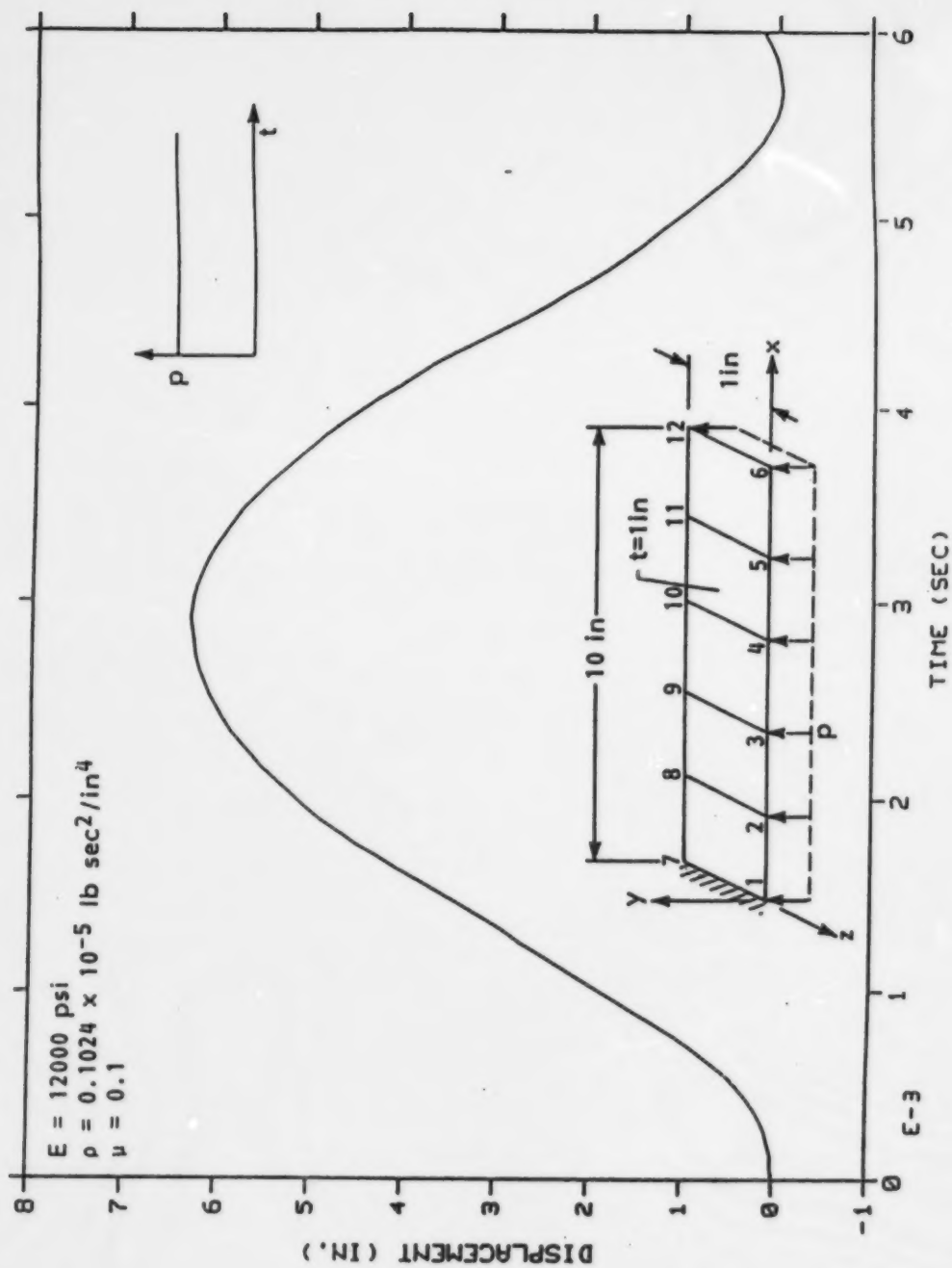
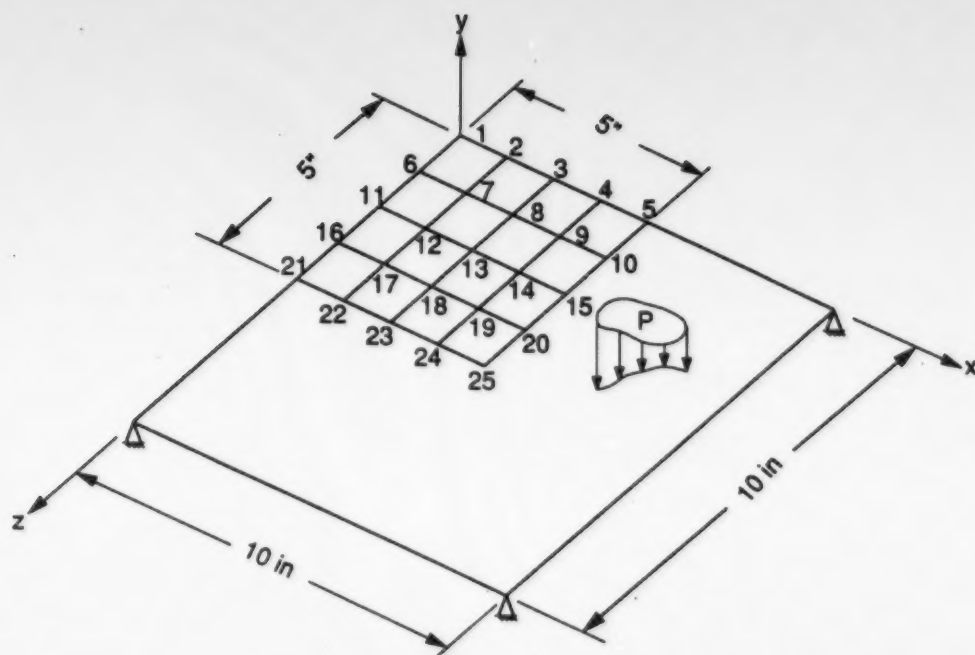


FIGURE 2.45.1

CASE 45: GEOMETRIC NON-LINEAR CANTILEVER SHELL TEST - $P = 2.85 \text{ psi}$
 MODEL AND DISPLACEMENT RESPONSE OF NODE 6 AND 12 IN THE Y-DIRECTION



FIGURE 2.45.2
CASE 45: GEOMETRIC NON-LINEAR CONTILEVER SHELL TEST - $P = 2.85$ psi
RESPONSE AT TIME $t = 4$ ms



Thickness = 0.5 in
 $\rho = 2.588 \times 10^{-4} \text{ lb-sec}^2/\text{in}^4$
 $E = 10^7 \text{ psi}$
 $\mu = 0.3$

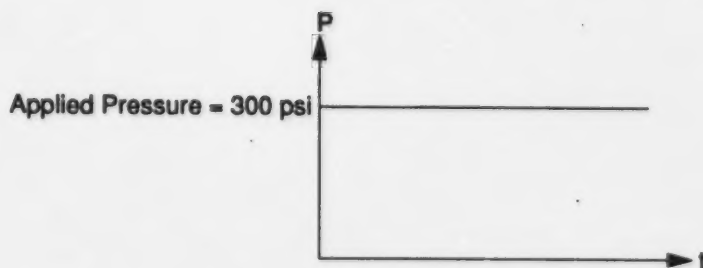
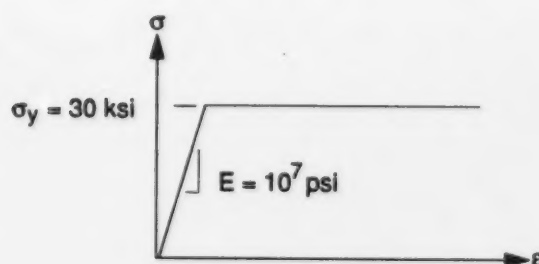


FIGURE 2.46.1

CASE 46: SIMPLY SUPPORTED SQUARE PLATE TEST
 MODEL, MATERIAL PROPERTIES AND APPLIED LOAD

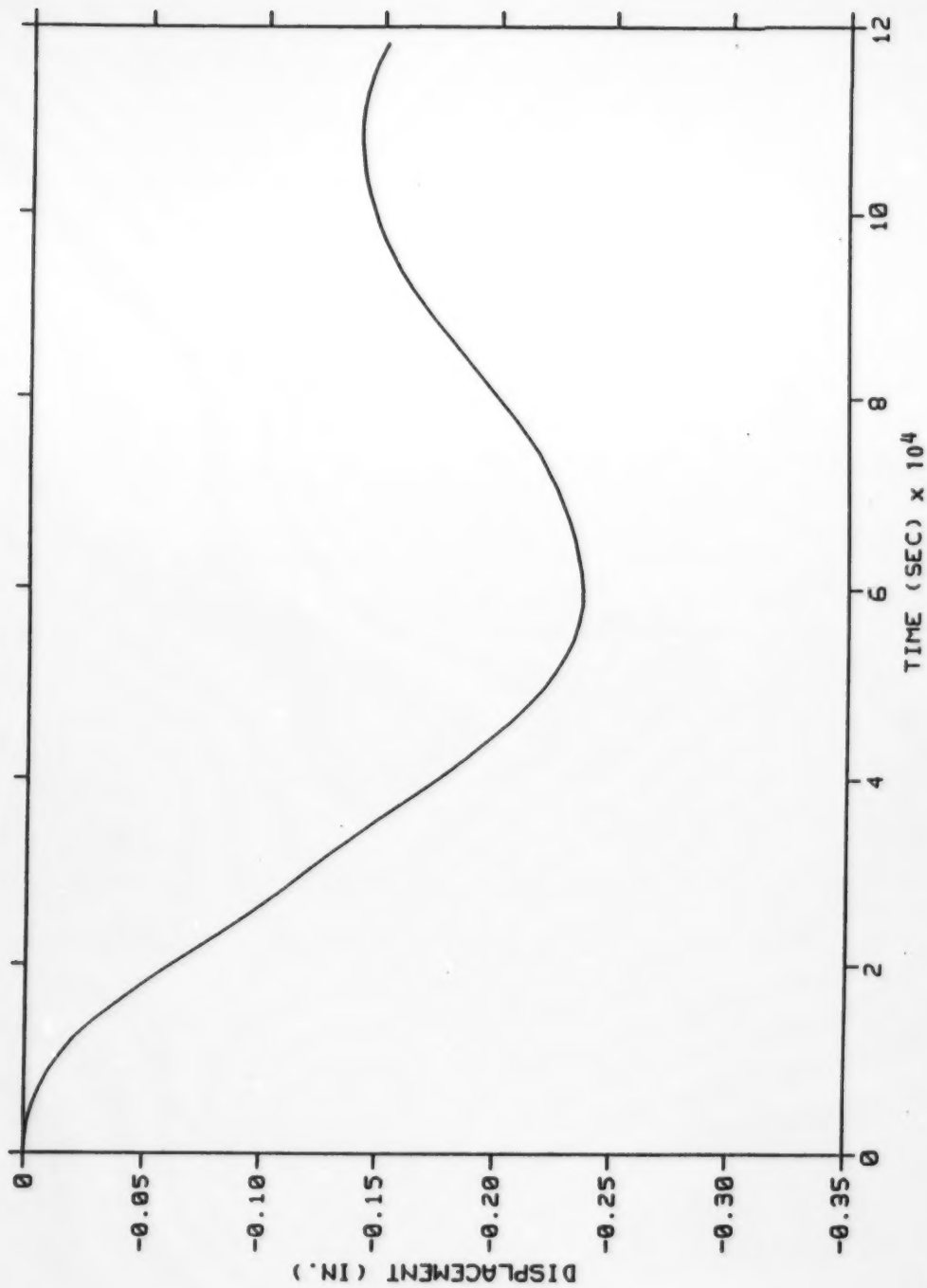


FIGURE 2.46.2
CASE 46: DISPLACEMENT AT NODE 25 IN THE Y-DIRECTION
ELASTIC-PLASTIC MATERIAL

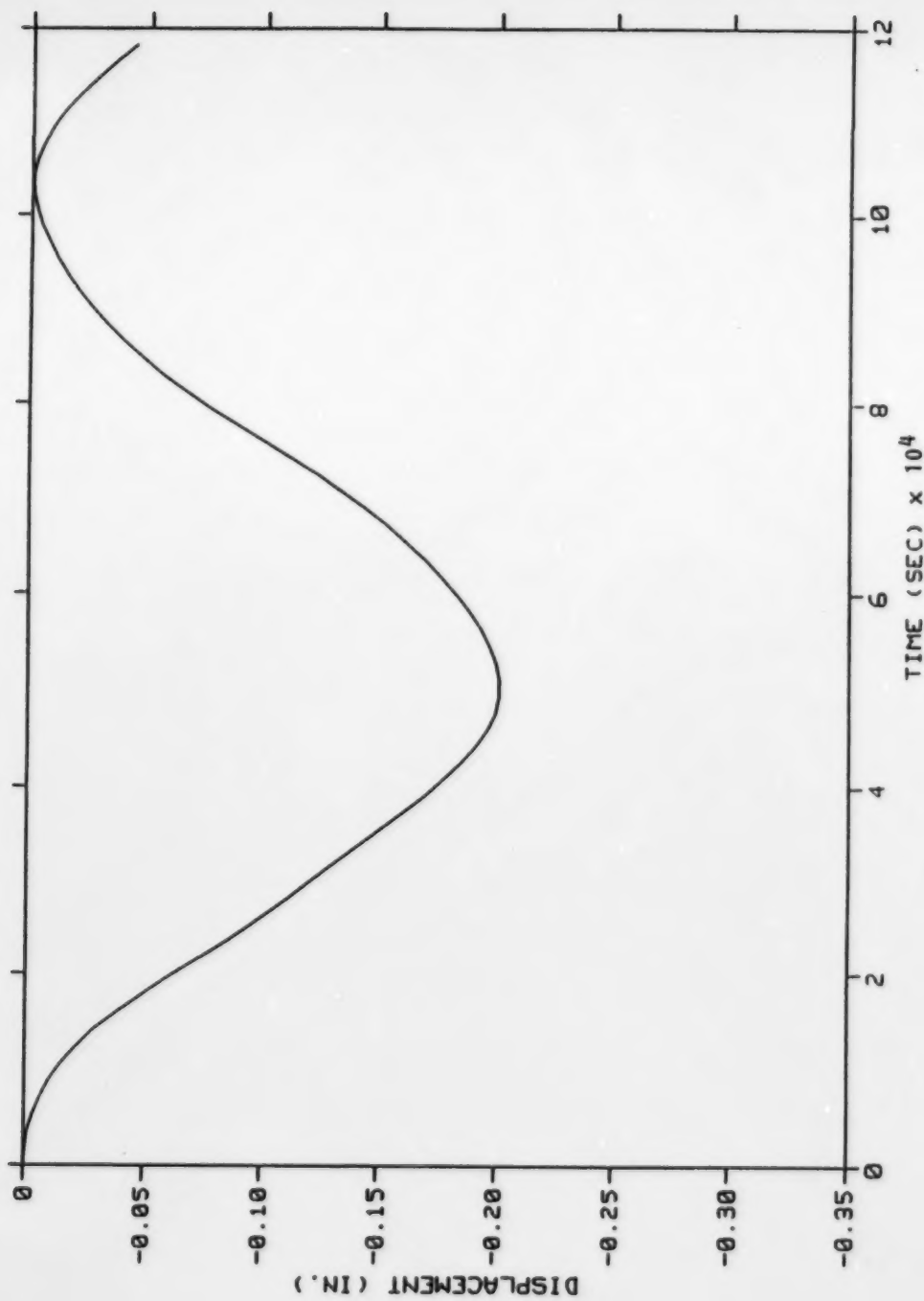


FIGURE 2.46.3
CASE 46: DISPLACEMENT AT NODE 25 IN THE Y-DIRECTION
ELASTIC MATERIAL

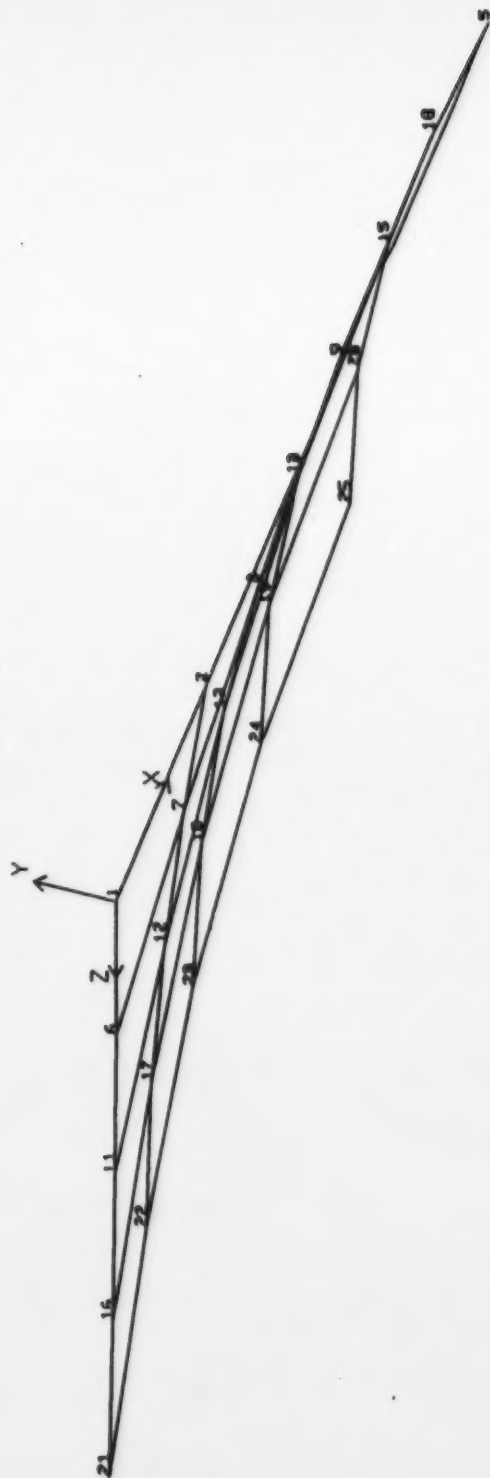


FIGURE 2.46.4
CASE 46: RESPONSE AT TIME = 0.7 ms

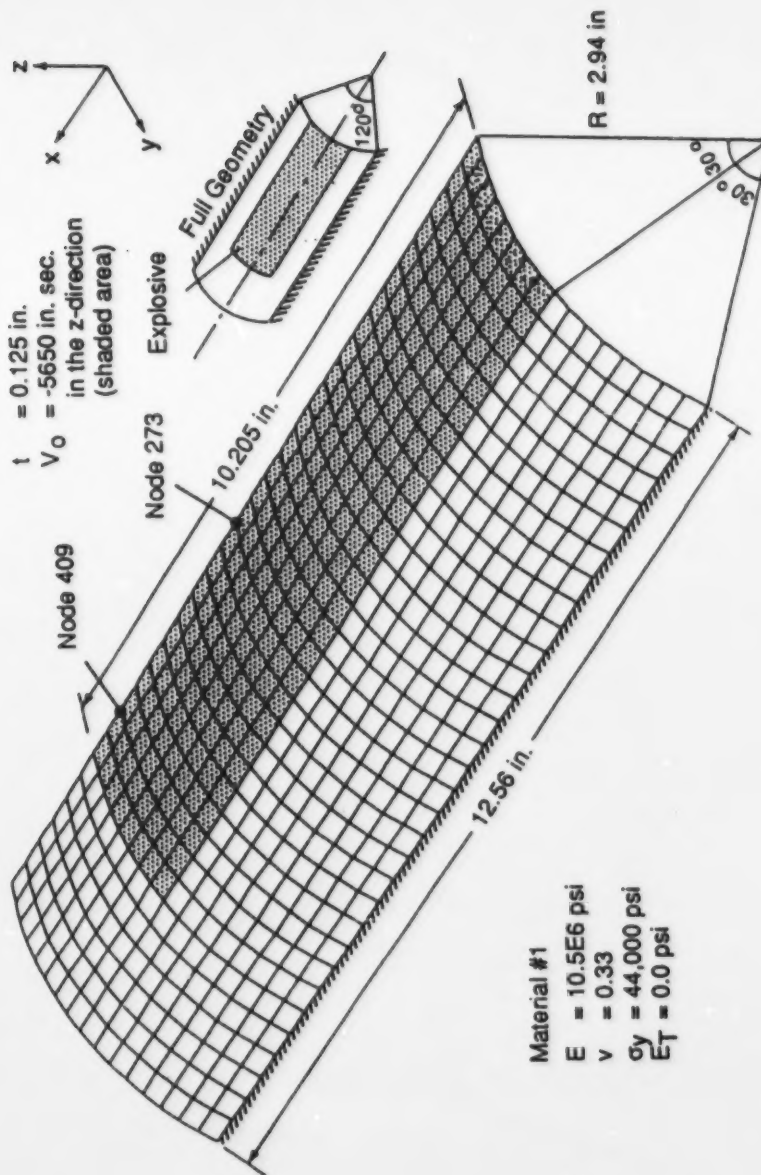


FIGURE 2.47.1

CASE 47: IMPULSIVELY LOADED CYLINDRICAL SHELL

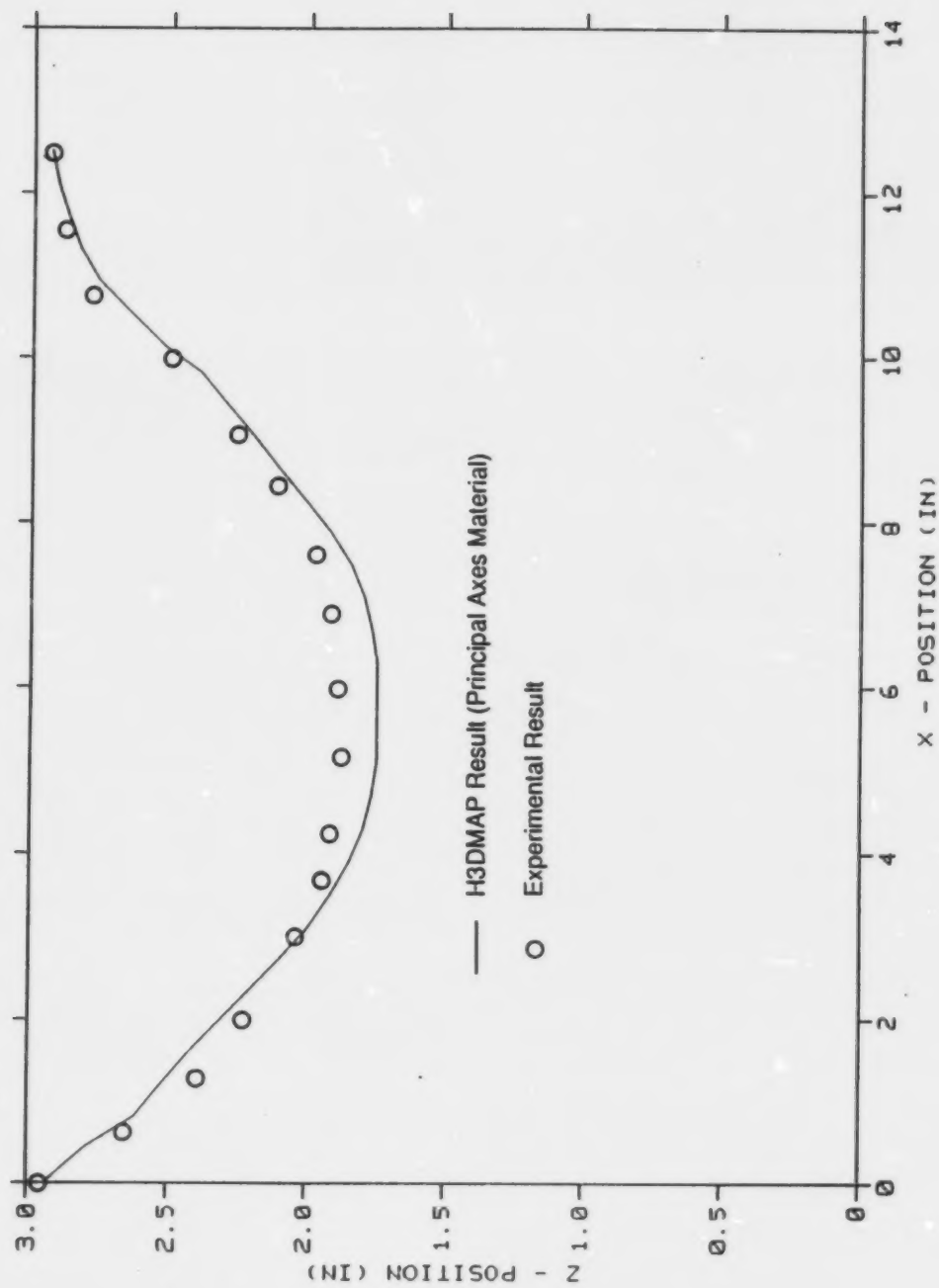


FIGURE 2.47.2
CASE 47: IMPULSIVELY LOADED CYLINDRICAL SHELL
DEFORMED SHAPE OF CROWN LINE

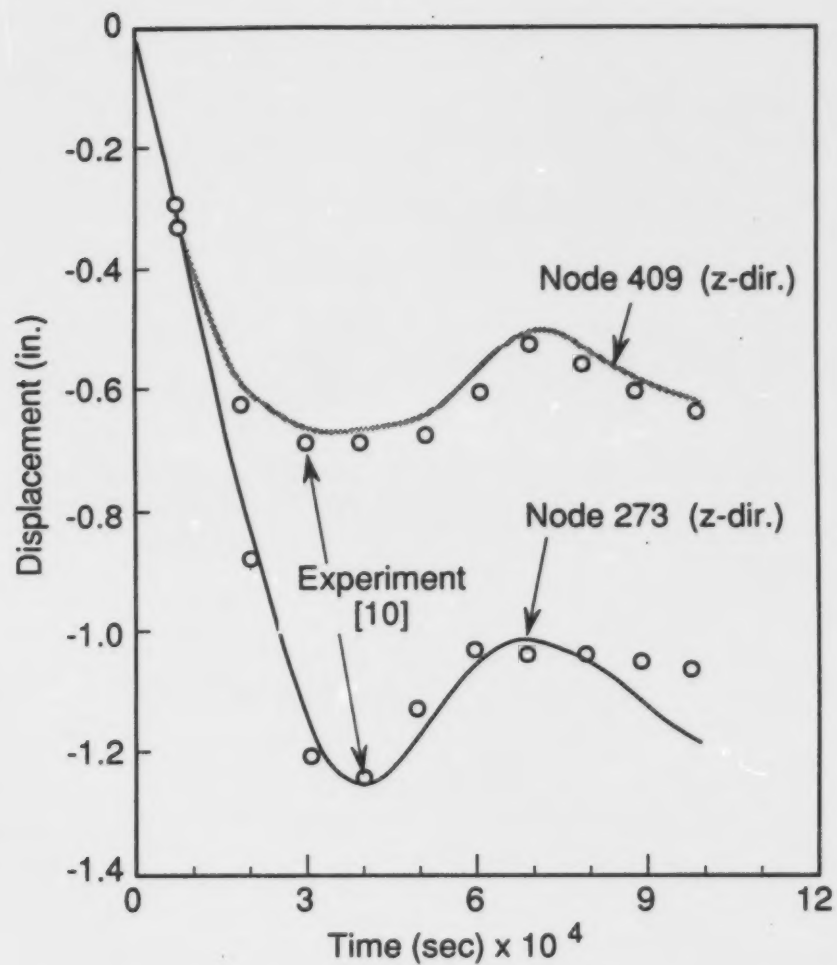


FIGURE 2.47.3

CASE 47: IMPULSIVELY LOADED CYLINDRICAL SHELL
DISPLACEMENT TIME HISTORY AT NODES 273, 409

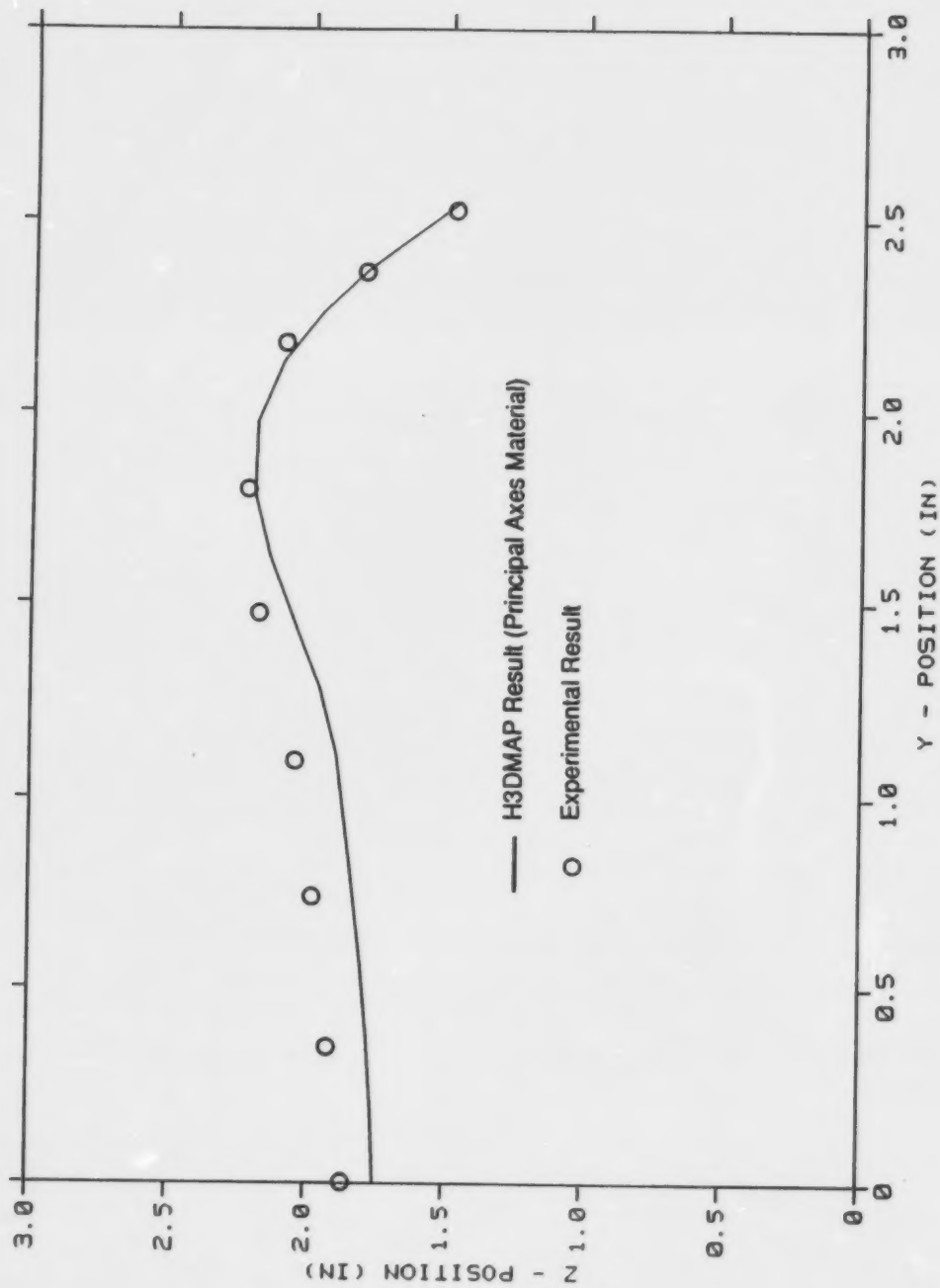


FIGURE 2.47.4
CASE 47: IMPULSIVELY LOADED CYLINDRICAL SHELL
DEFORMED SHAPE AT $X = 6.28$

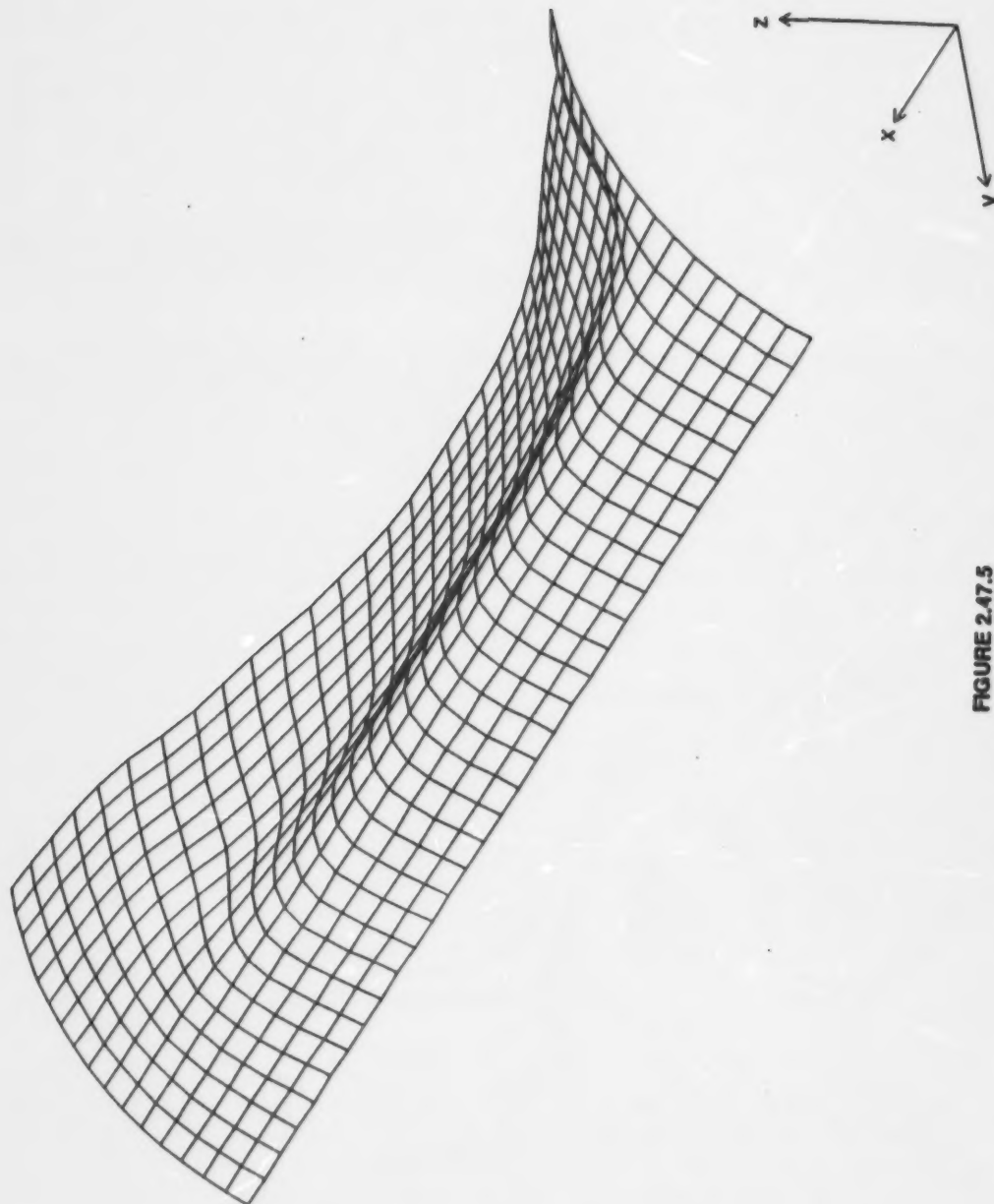
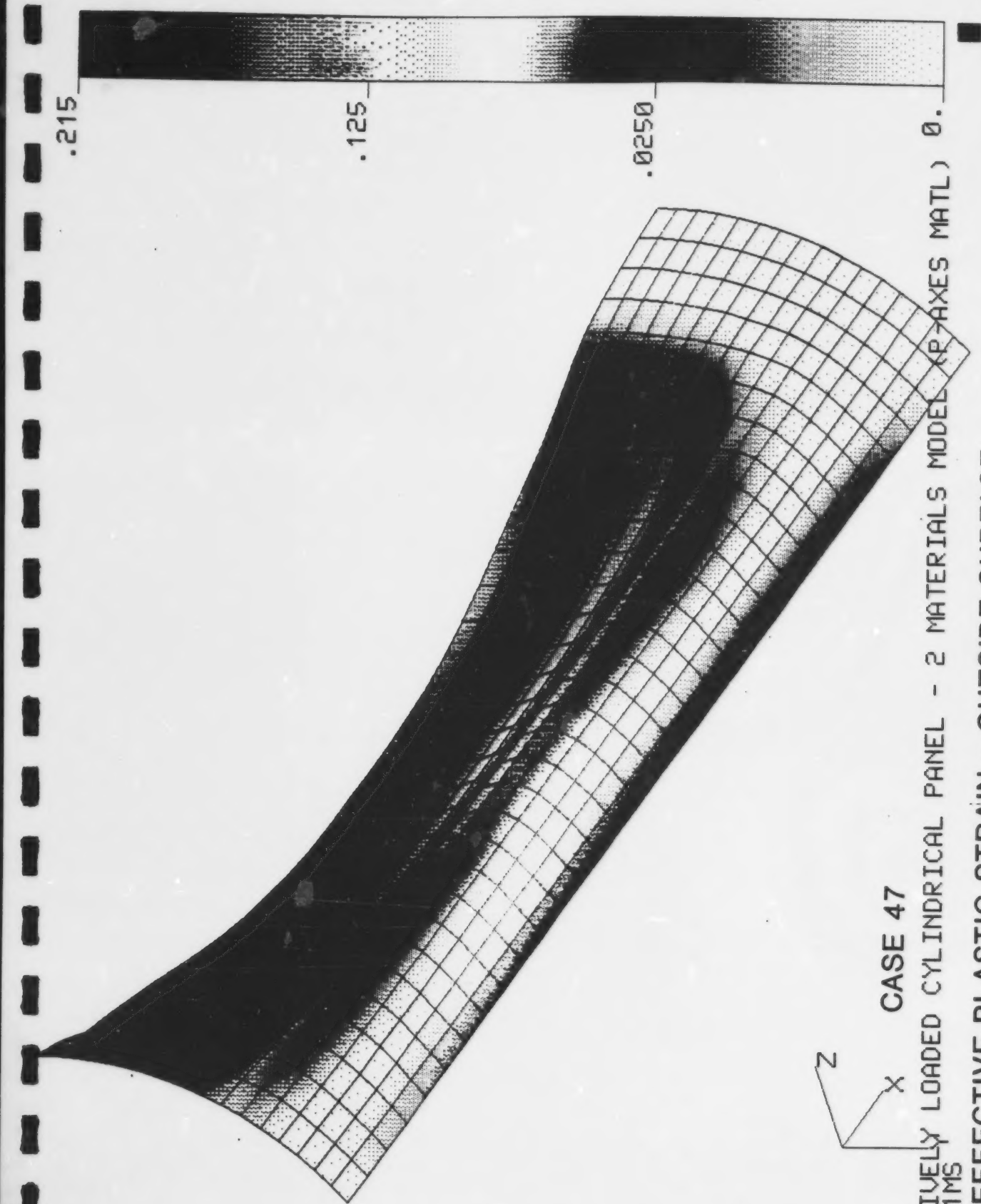


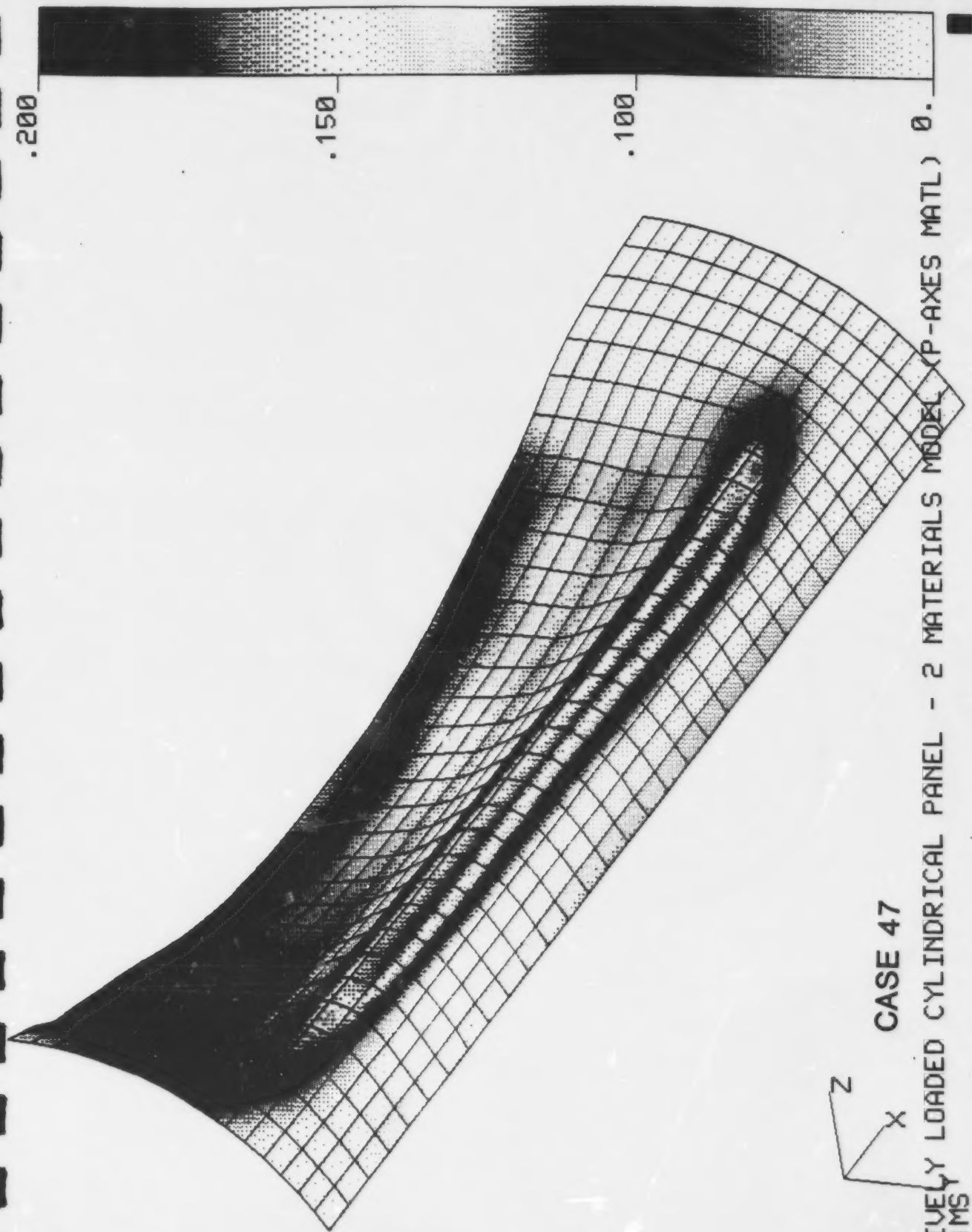
FIGURE 2.47.5
CASE 47: IMPULSIVELY LOADED CYLINDRICAL SHELL
RESPONSE AT TIME = 0.001 s



IMPULSIVELY LOADED CYLINDRICAL PANEL - 2 MATERIALS MODEL (P-AXES MATL) $t = 1\text{ms}$

EFFECTIVE PLASTIC STRAIN - OUTSIDE SURFACE

FIGURE 2.47.6

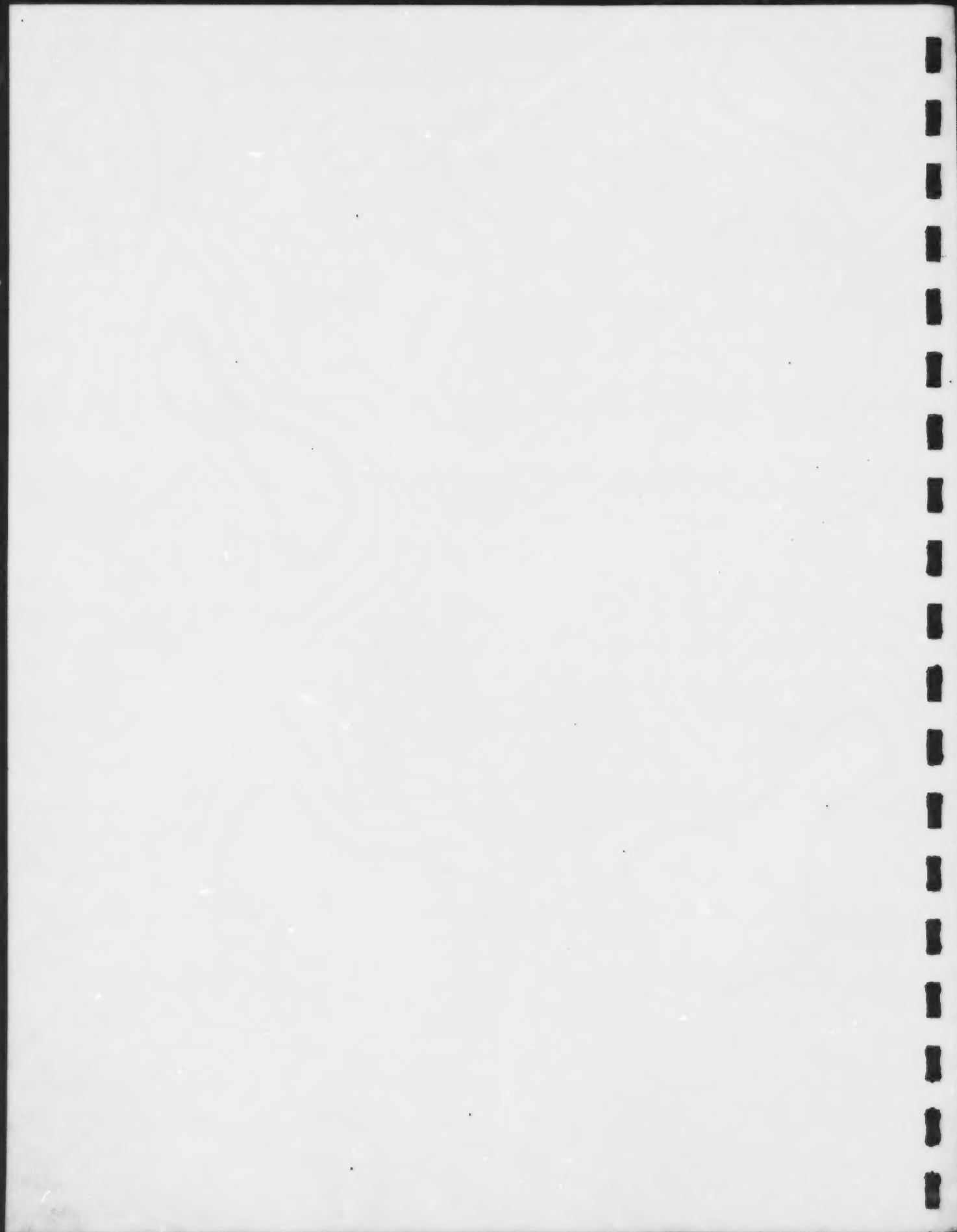


CASE 47

IMPULSIVELY LOADED CYLINDRICAL PANEL - 2 MATERIALS MODEL (P-AXES MODEL)
TIME = 1MS

EFFECTIVE PLASTIC STRAIN - INSIDE SURFACE

FIGURE 3.1.1



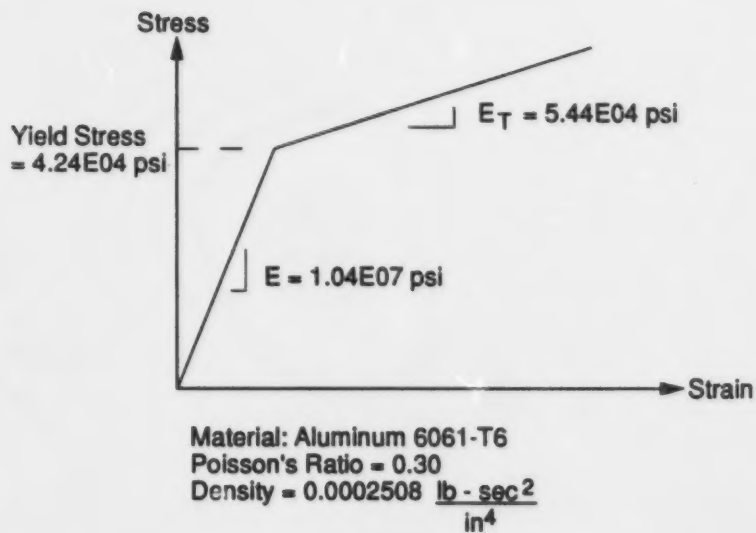
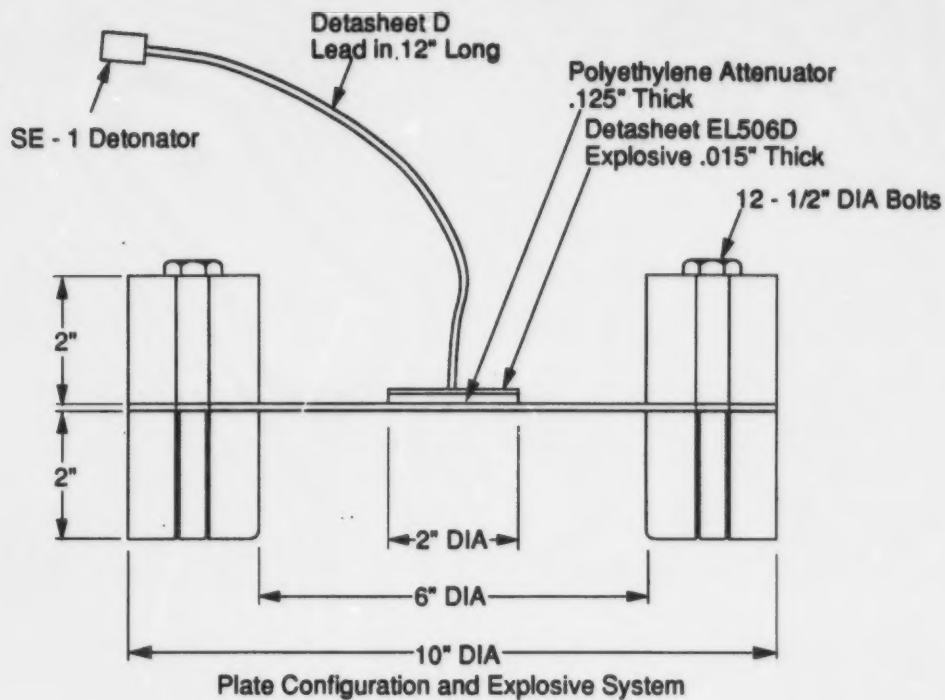


FIGURE 2.48.1

CASE 48: EXPERIMENTAL TEST RIG AND MATERIAL PROPERTIES

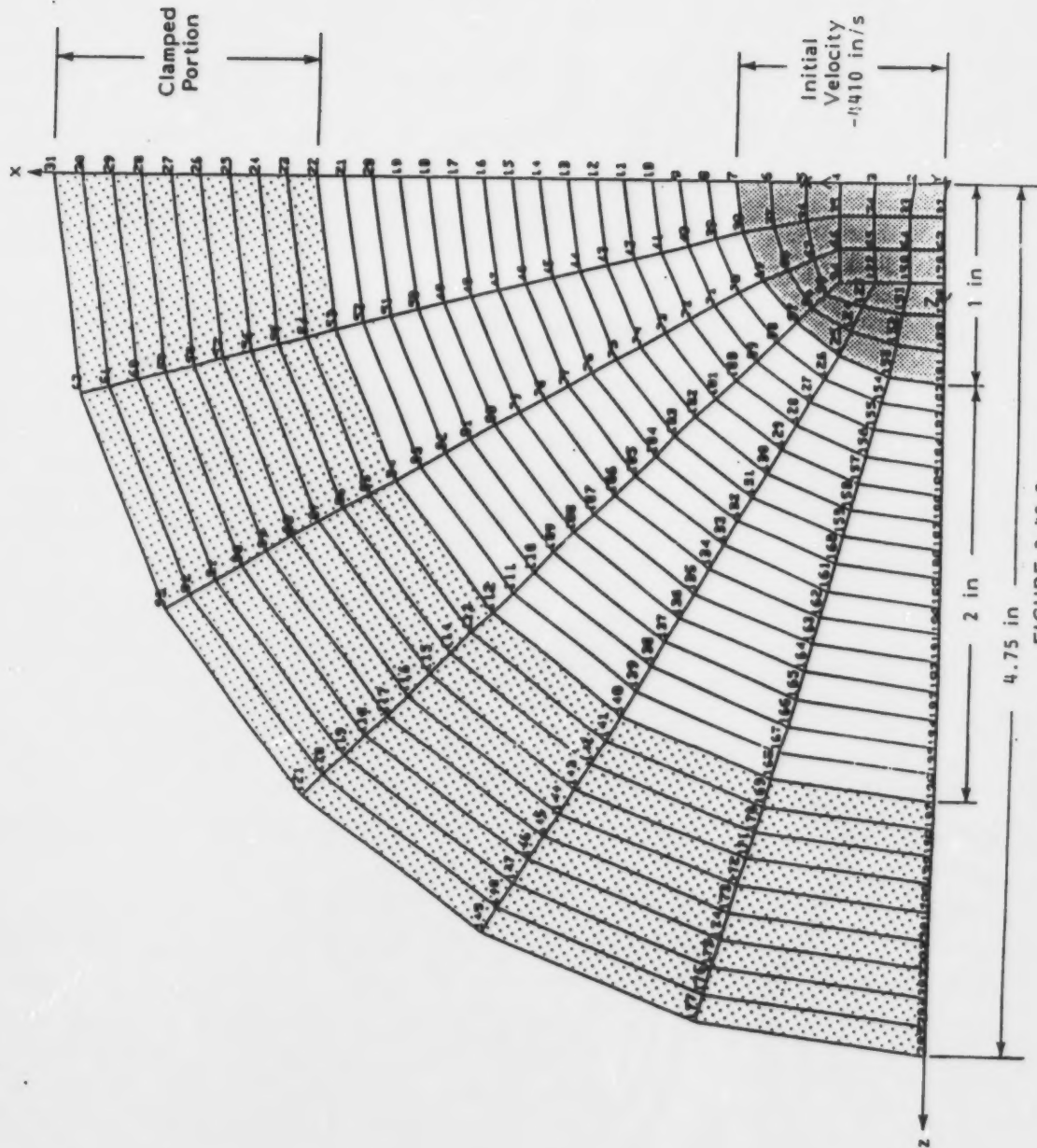


FIGURE 2.48.2
CASE 48: IMPULSIVELY LOADED CIRCULAR PLATE - NODE NUMBER

181642-RD

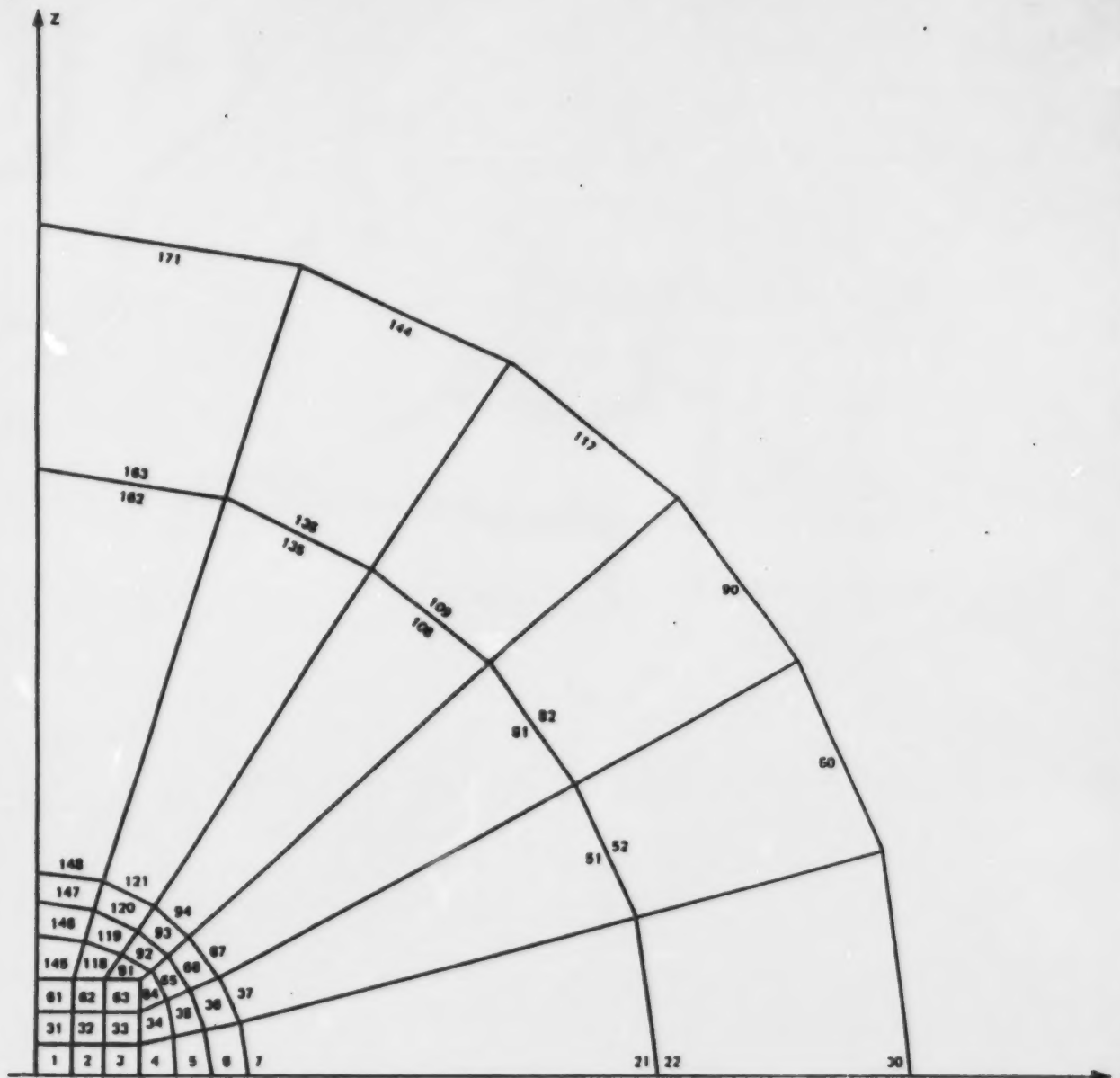


FIGURE 2.48.3
CASE 48: IMPULSIVELY LOADED CIRCULAR PLATE
ELEMENT NUMBERS

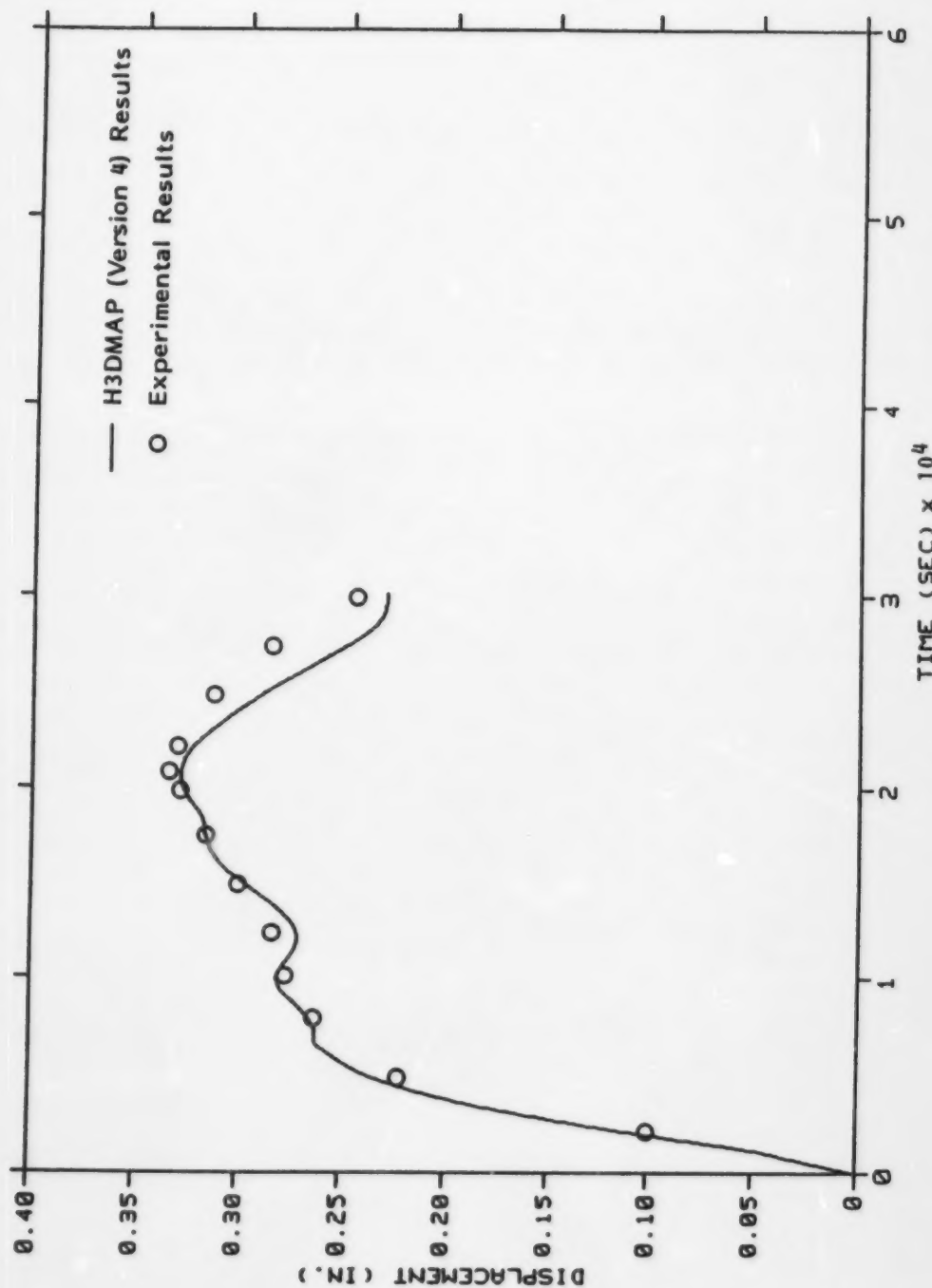


FIGURE 2.48.4
CASE 48: DISPLACEMENT IN Y-DIRECTION AT NODE 1
COMPARISON OF THEORETICAL AND EXPERIMENTAL RESULTS



FIGURE 2.48.5
CASE 48: RESPONSE AT TIME = 0.2 ms

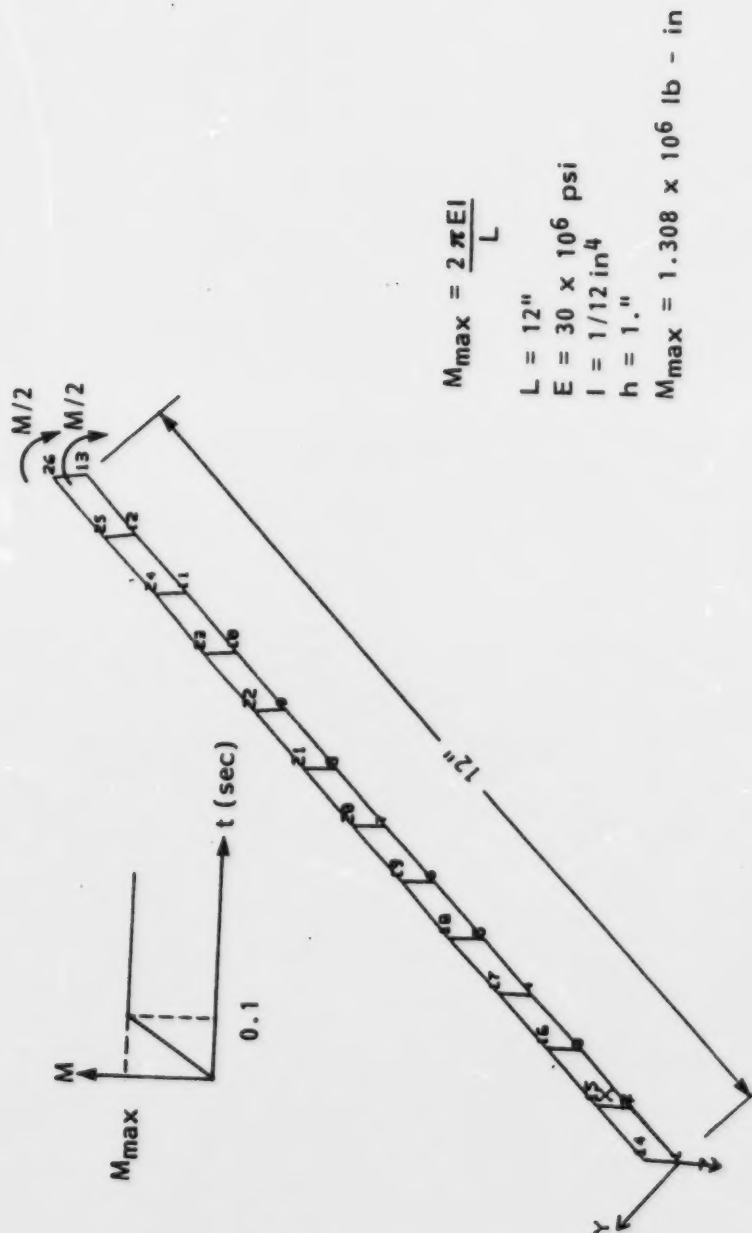


FIGURE 2.49.1
CASE 49: MOMENT LOAD TEST FOR NON-LINEAR SHELL (DYNAMIC RELAXATION)
MODEL AND LOAD

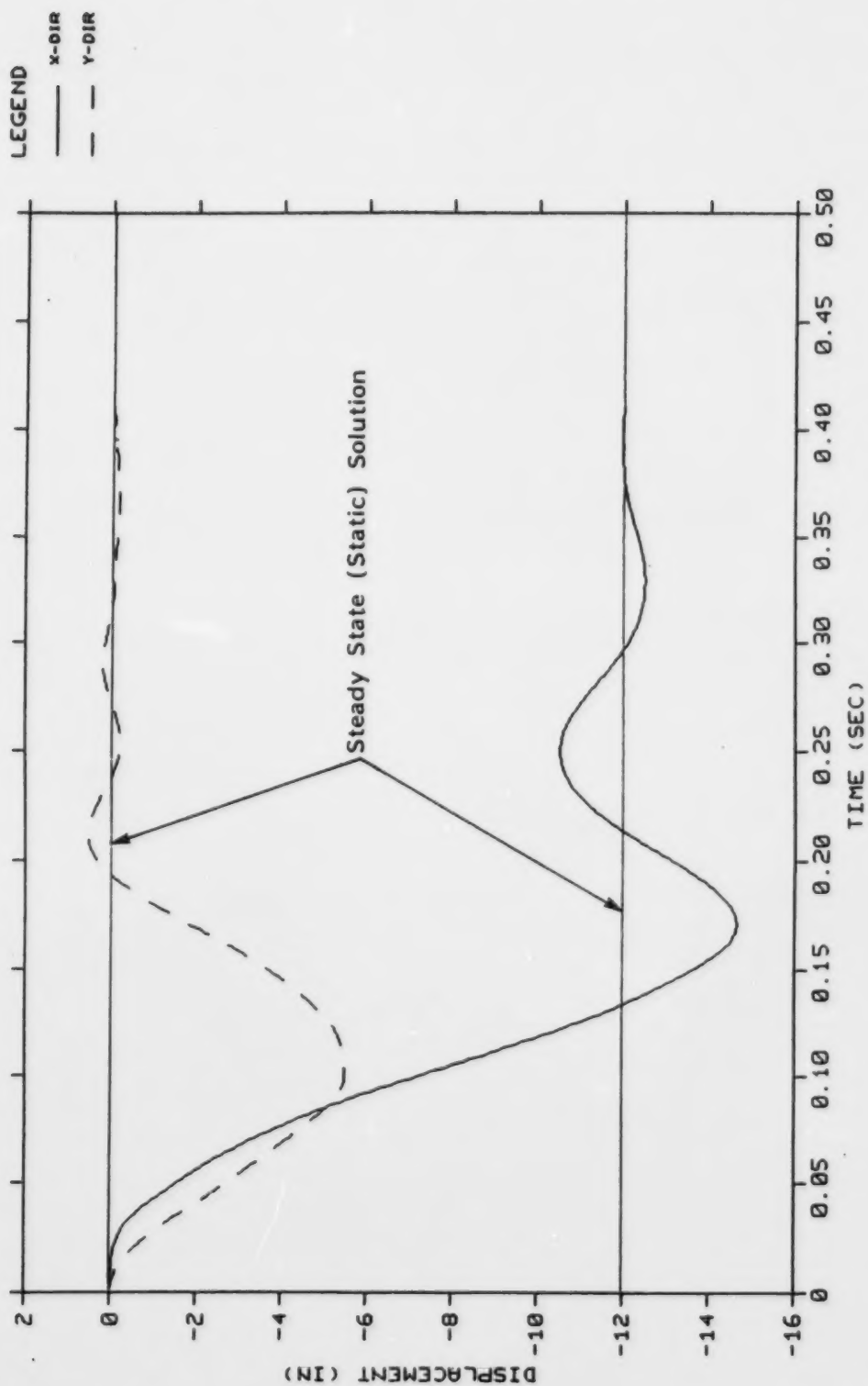


FIGURE 2.49.2

CASE 49: DISPLACEMENT OF NODE 13 IN BOTH X AND Y DIRECTIONS

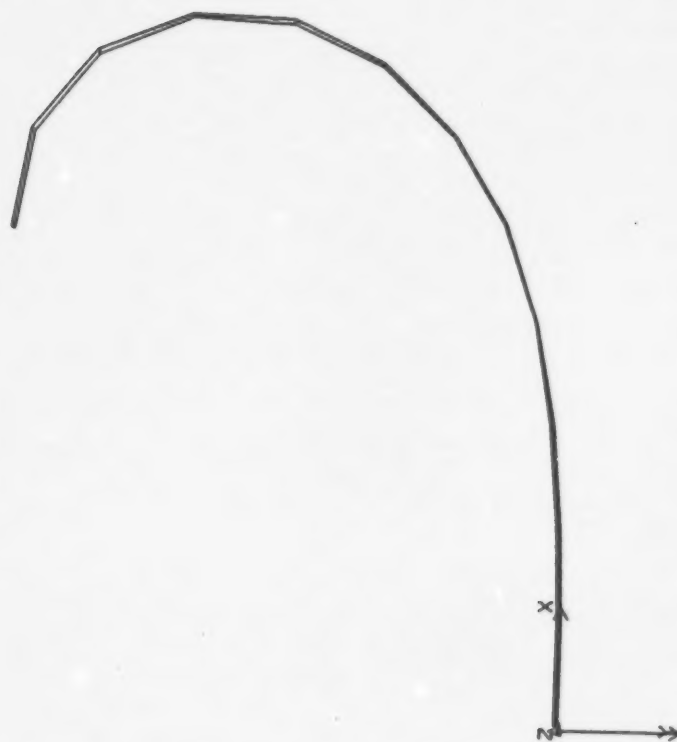
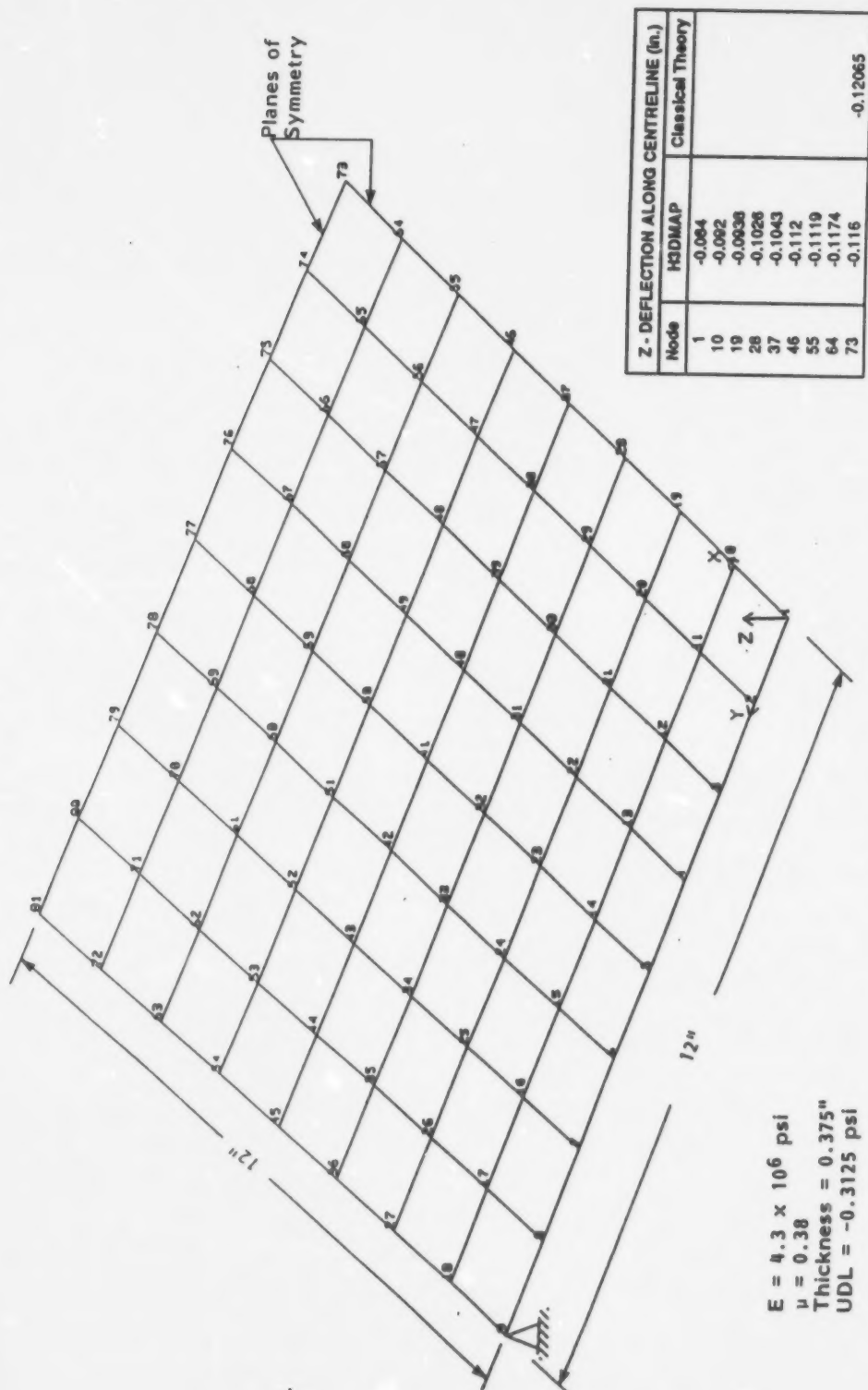


FIGURE 2.49.3
CASE 49: RESPONSE AT TIME = 0.1 sec



FIGURE 2.49.4
CASE 49: RESPONSE AT TIME = 0.4 sec



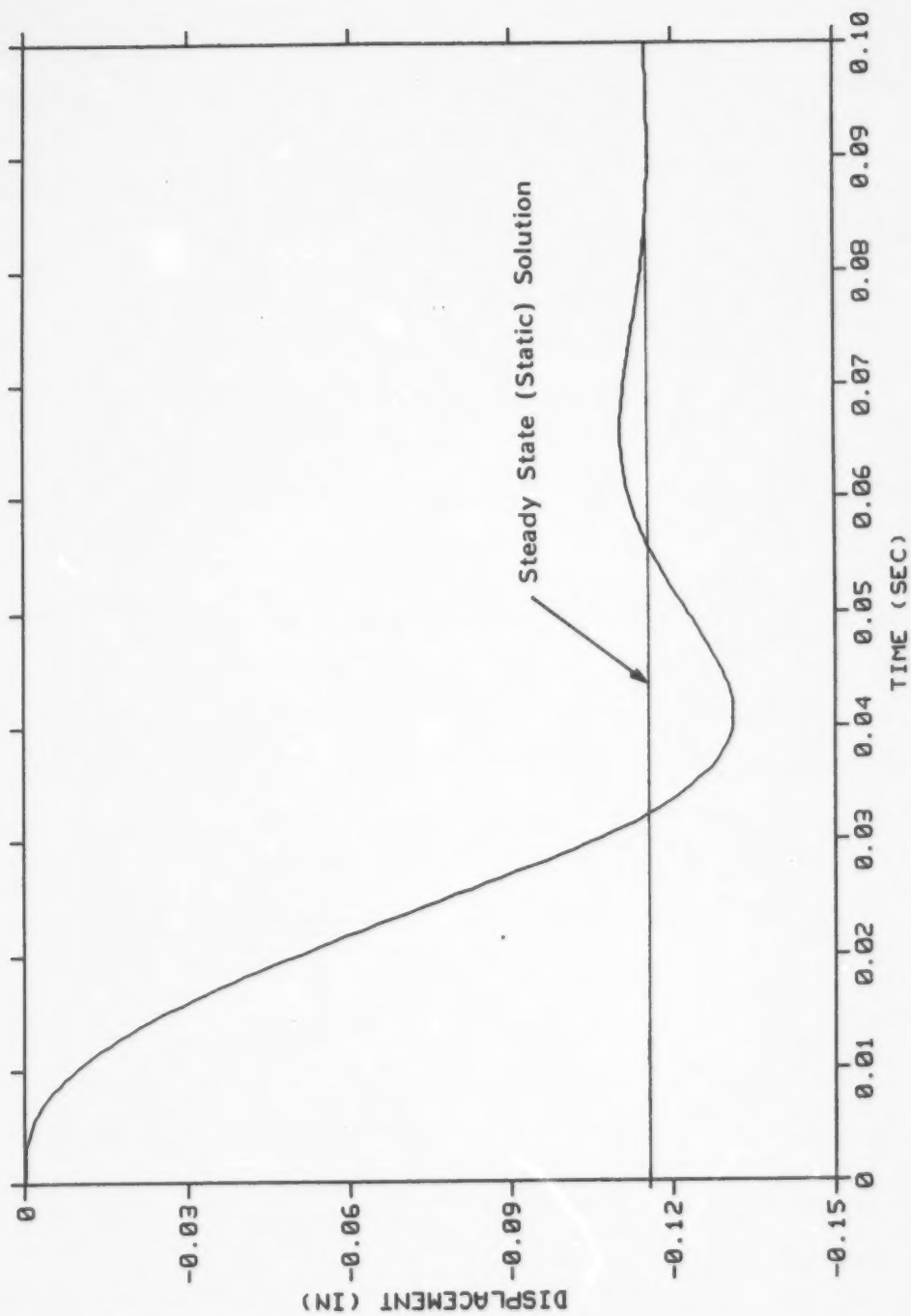


FIGURE 2.50.2
CASE 50: VERTICAL DEFLECTION OF NODE 73

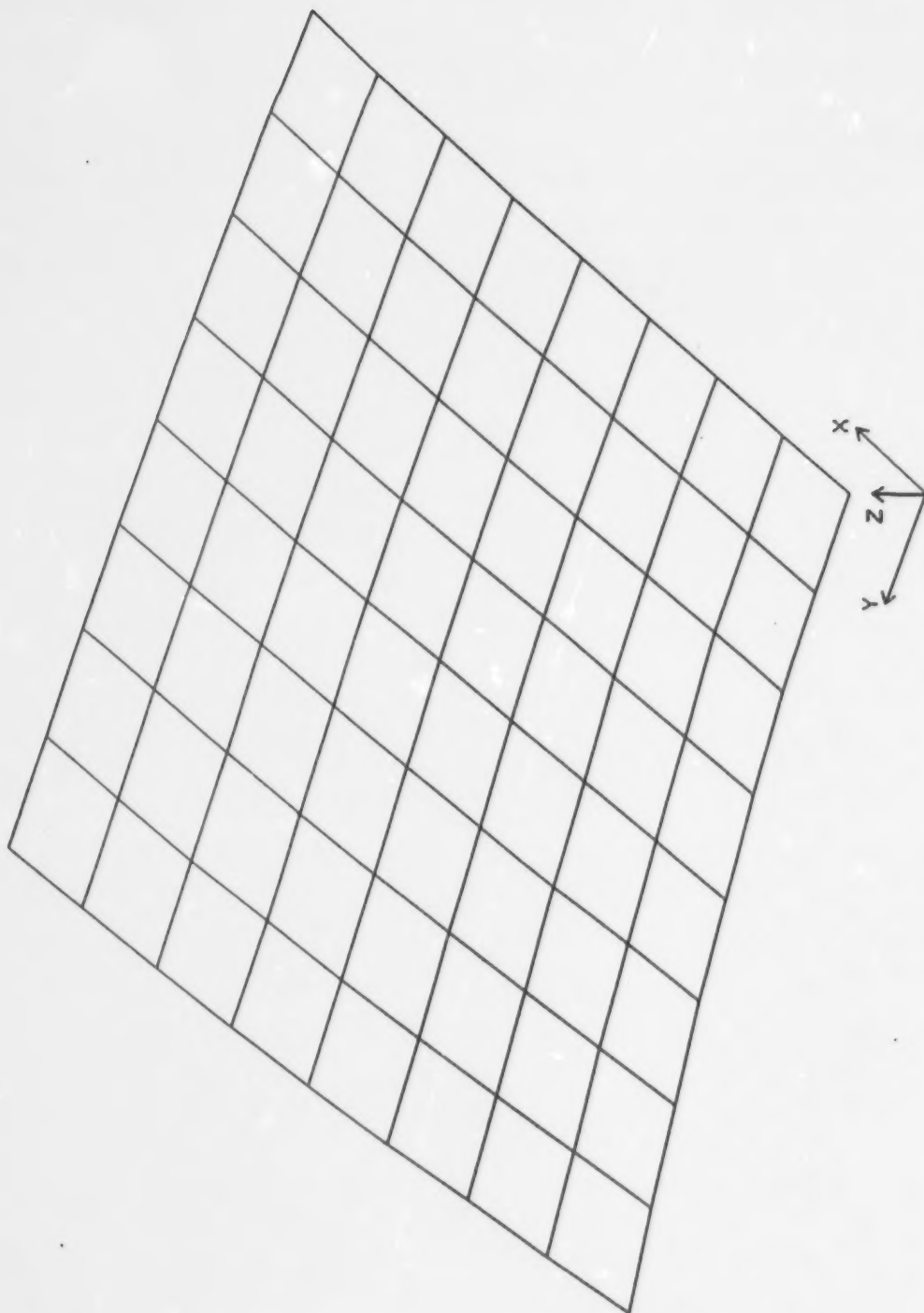


FIGURE 2.50.3
CASE 50: RESPONSE AT TIME = 0.02 sec

SOLDEX1 - Node 104

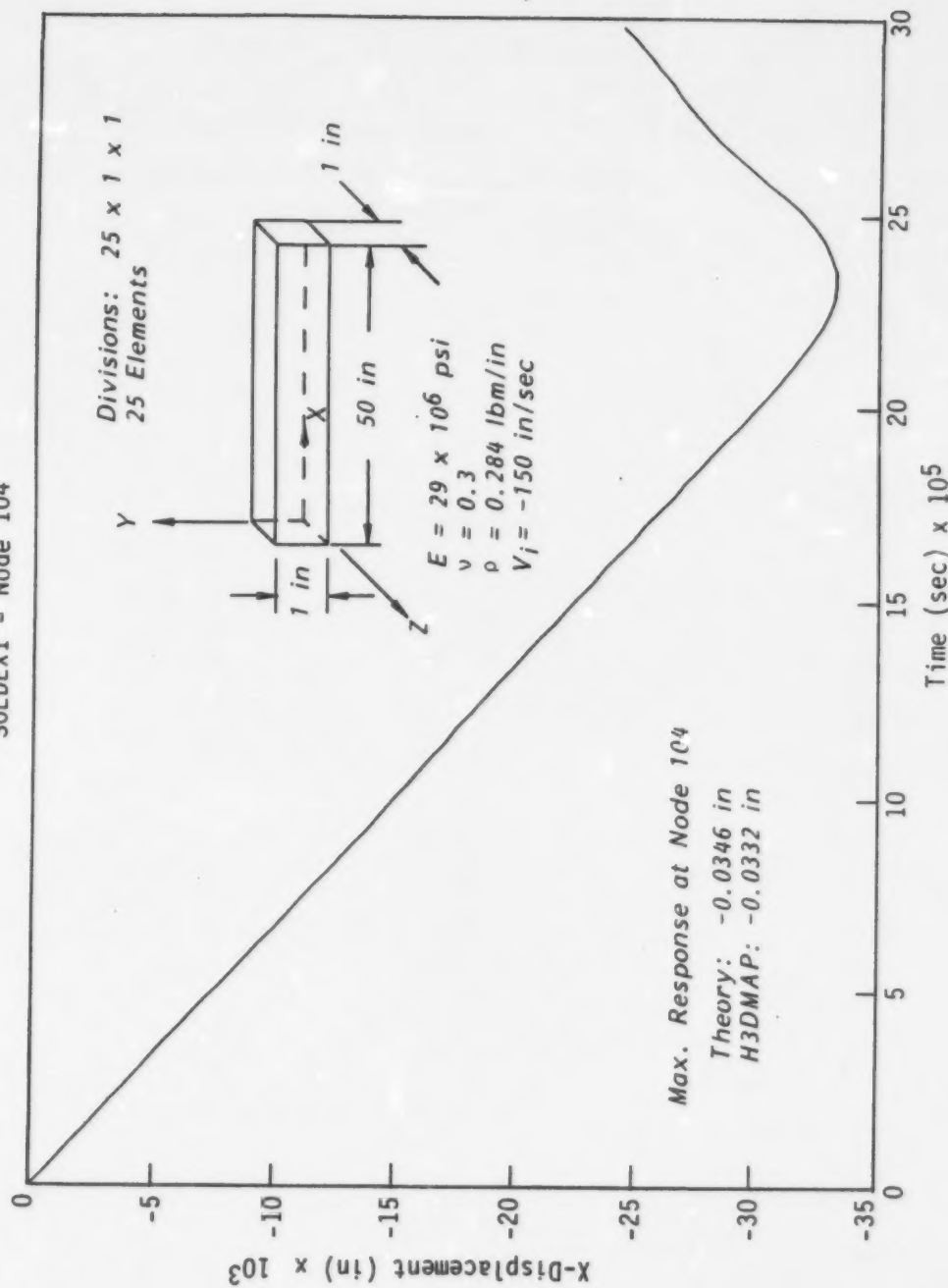


FIGURE 2.51.1
 CASE 51: AXIAL IMPACT OF A 3-D BEAM - RIGID WALL
 MODEL AND RESPONSE OF NODE 104

SOLDEX2 - Node 104

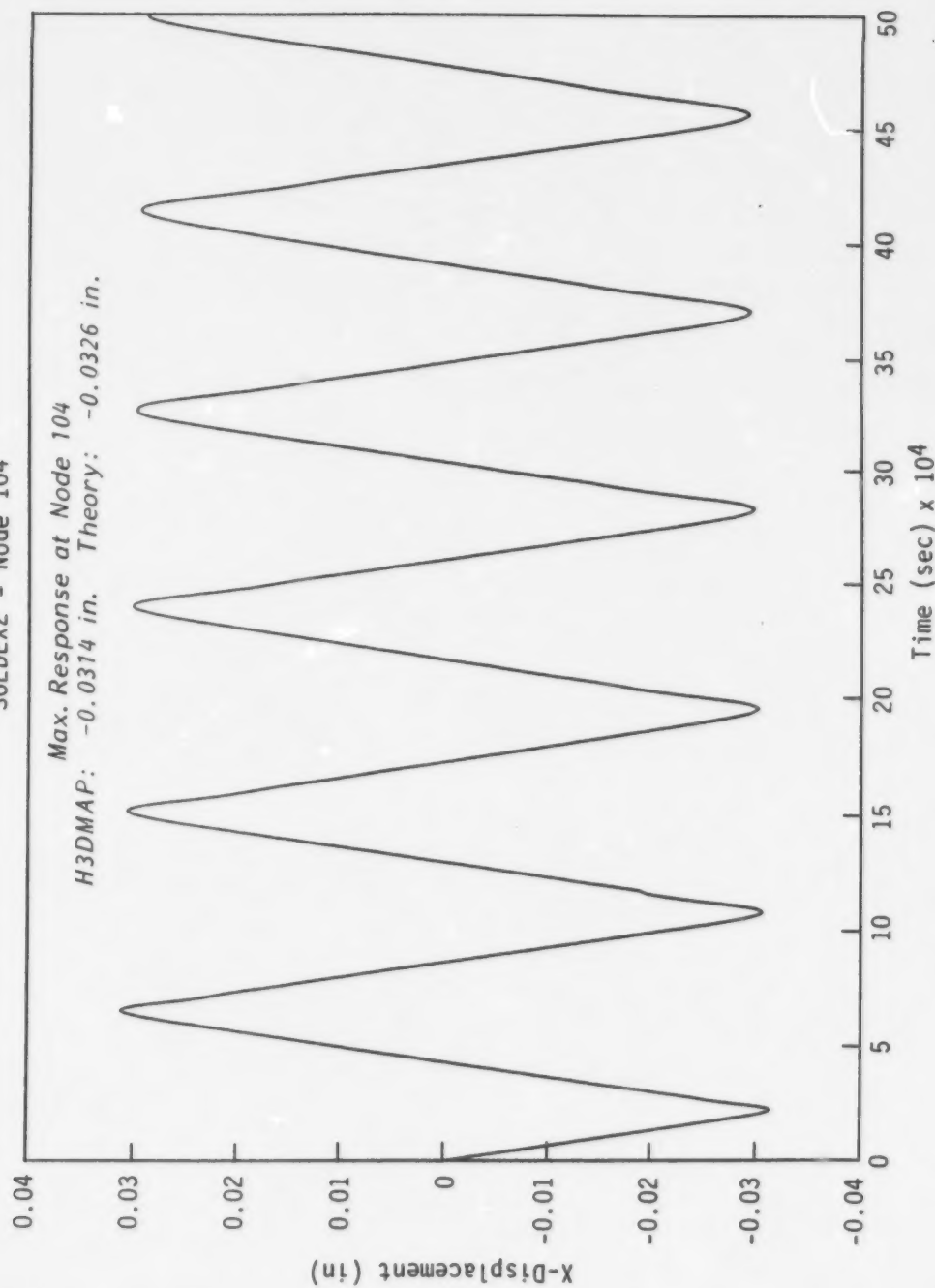


FIGURE 2.52.1
CASE 52: AXIAL IMPACT OF A 3-D BEAM - BOUNDARY RESTRICTIONS
RESPONSE OF NODE 104

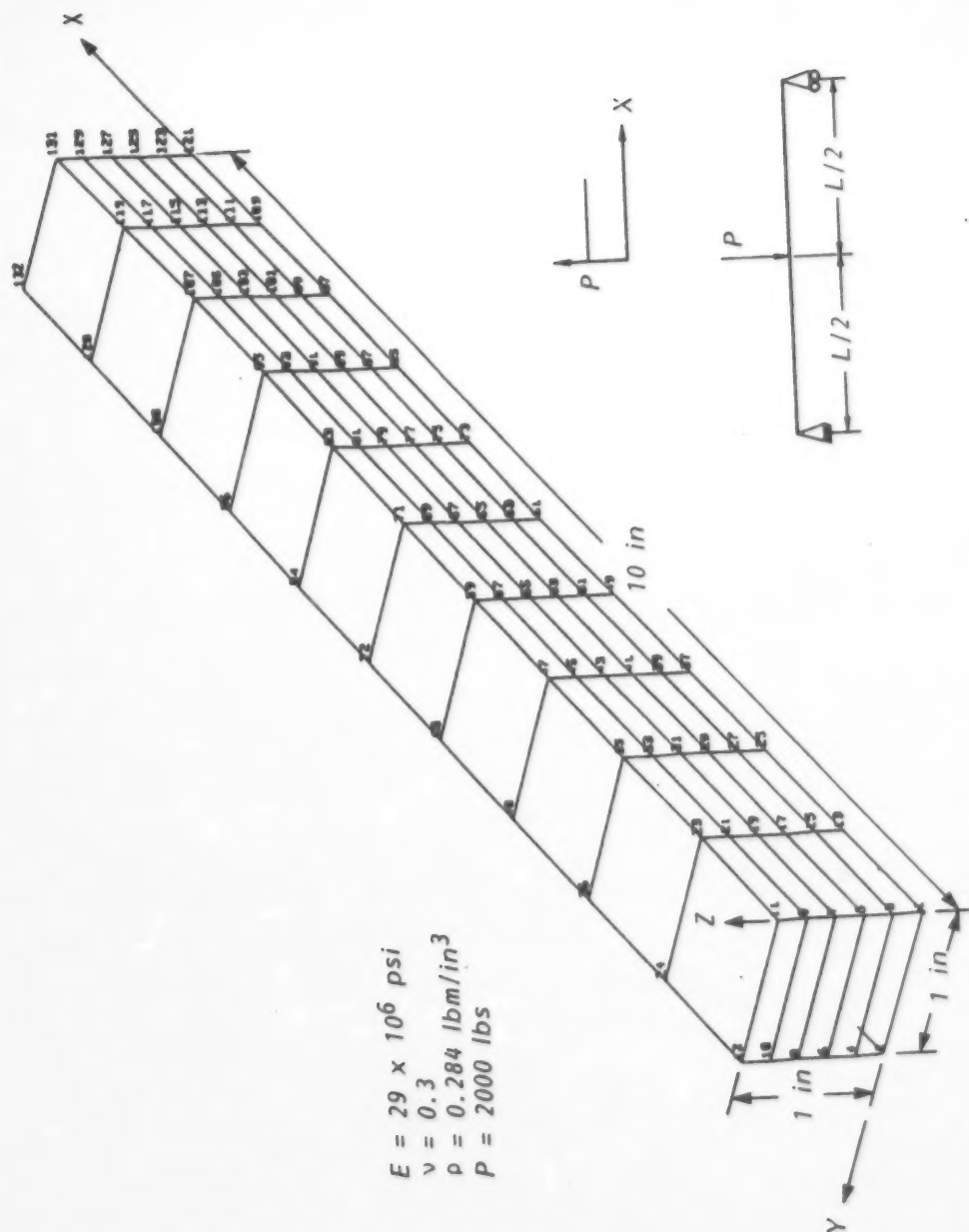


FIGURE 2.53.1
CASE 53: SIMPLY SUPPORTED BEAM MODEL
STEP LOAD AT MID-SPAN

SOLDEX3 - Node 72

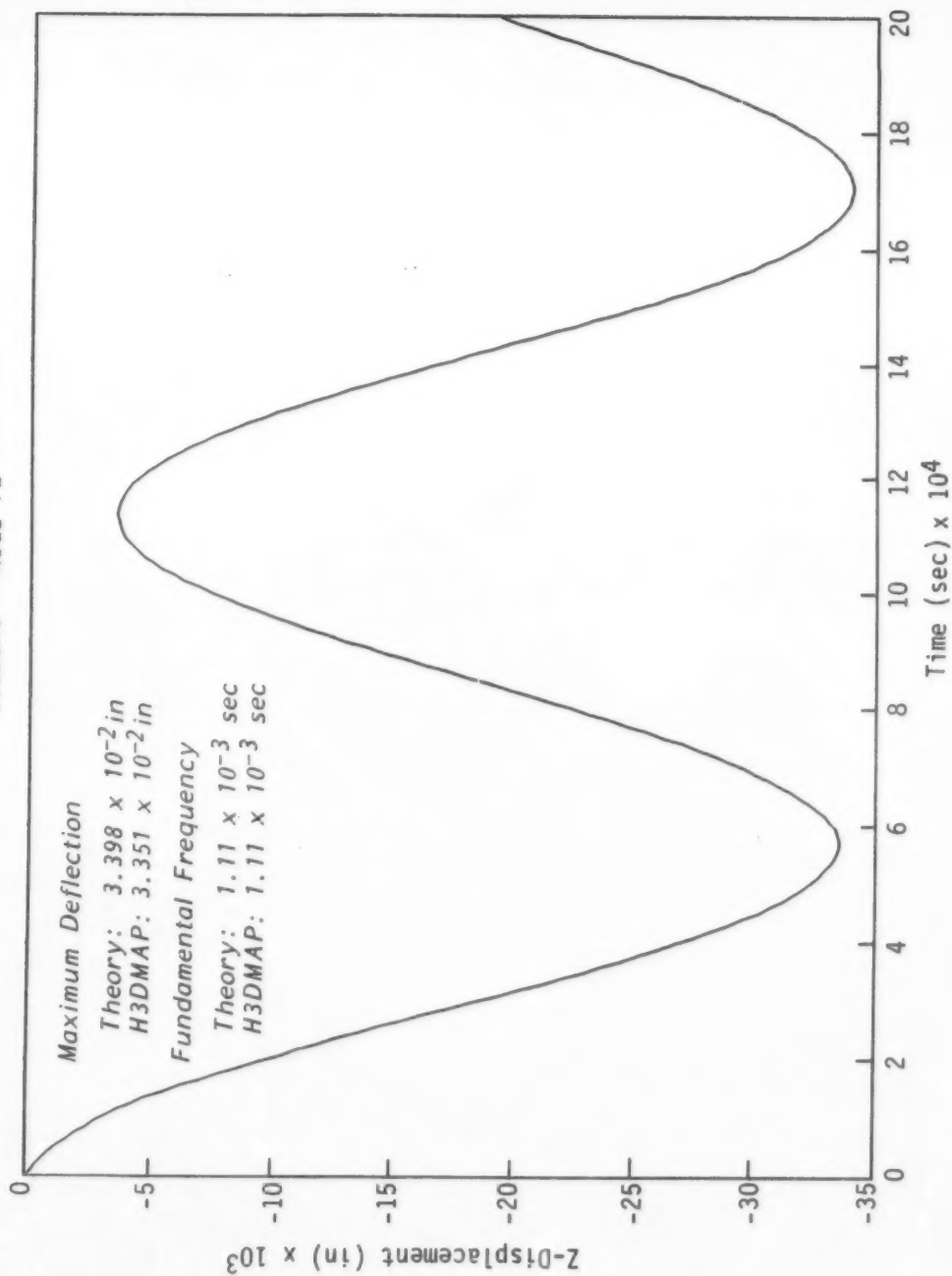


FIGURE 2.53.2
 CASE 53: SIMPLY SUPPORTED BEAM - STEP LOAD AT MID-SPAN
 RESPONSE OF NODE 72

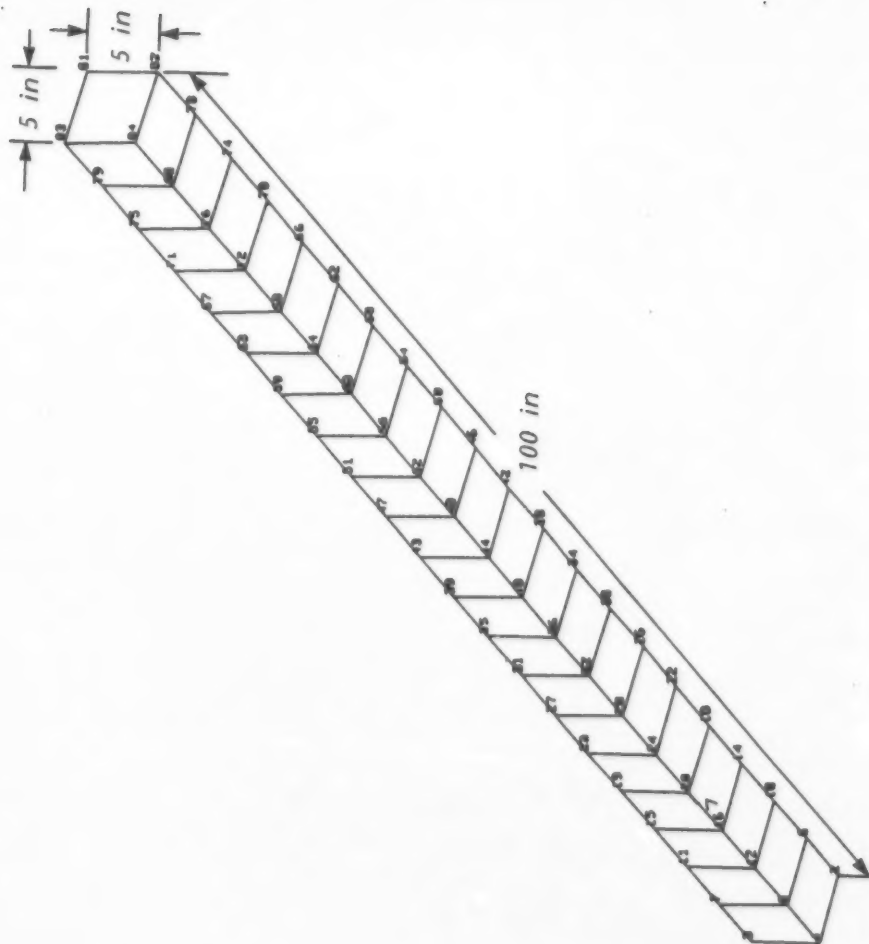


FIGURE 2.54.1
CASE 54: BEAM MODEL

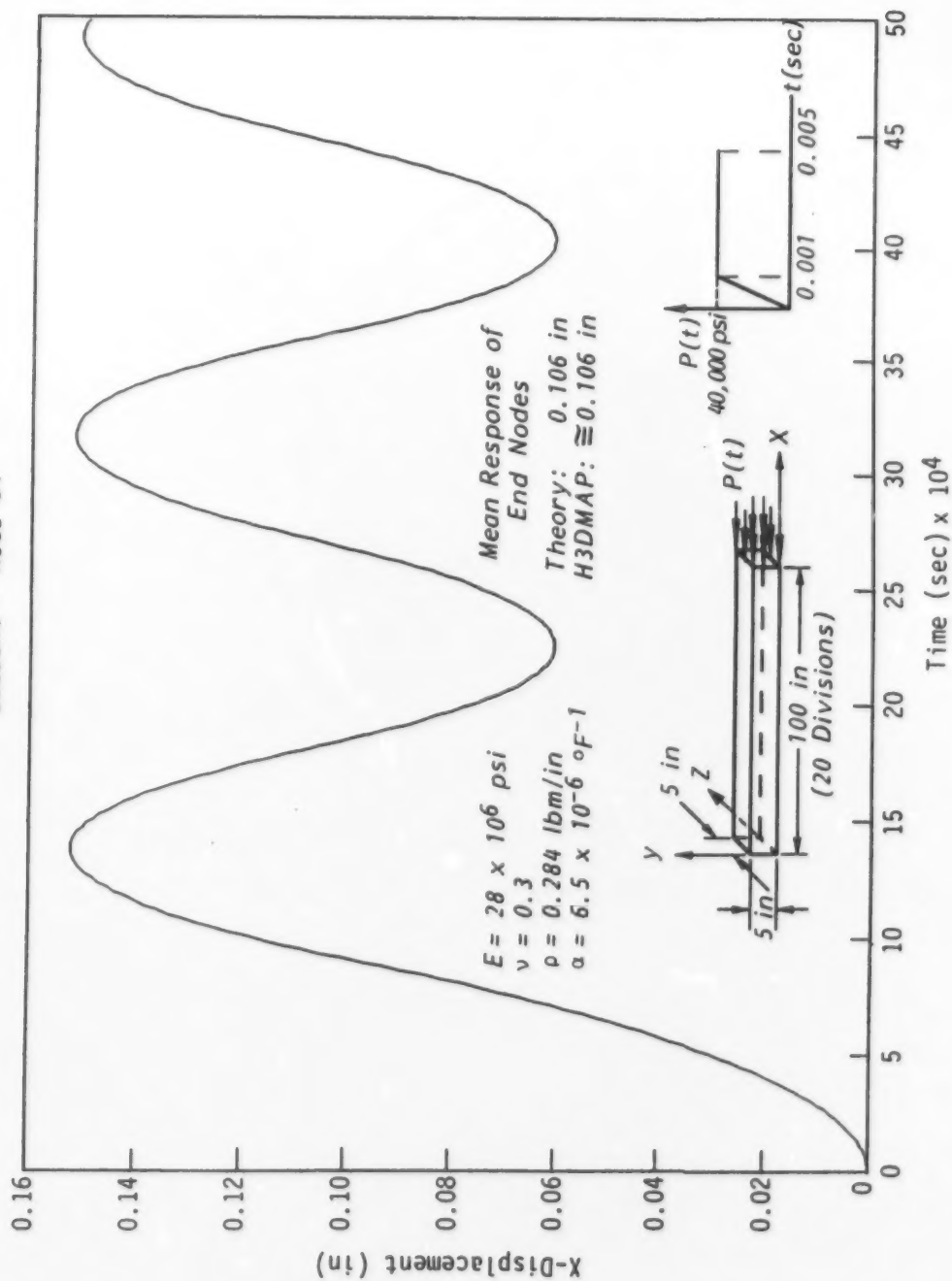


FIGURE 2.54.2

CASE 54: AXIAL END LOAD ON CANTILEVER BEAM
MODEL AND RESPONSE OF NODE 84

SOLDEX6 - Node 84

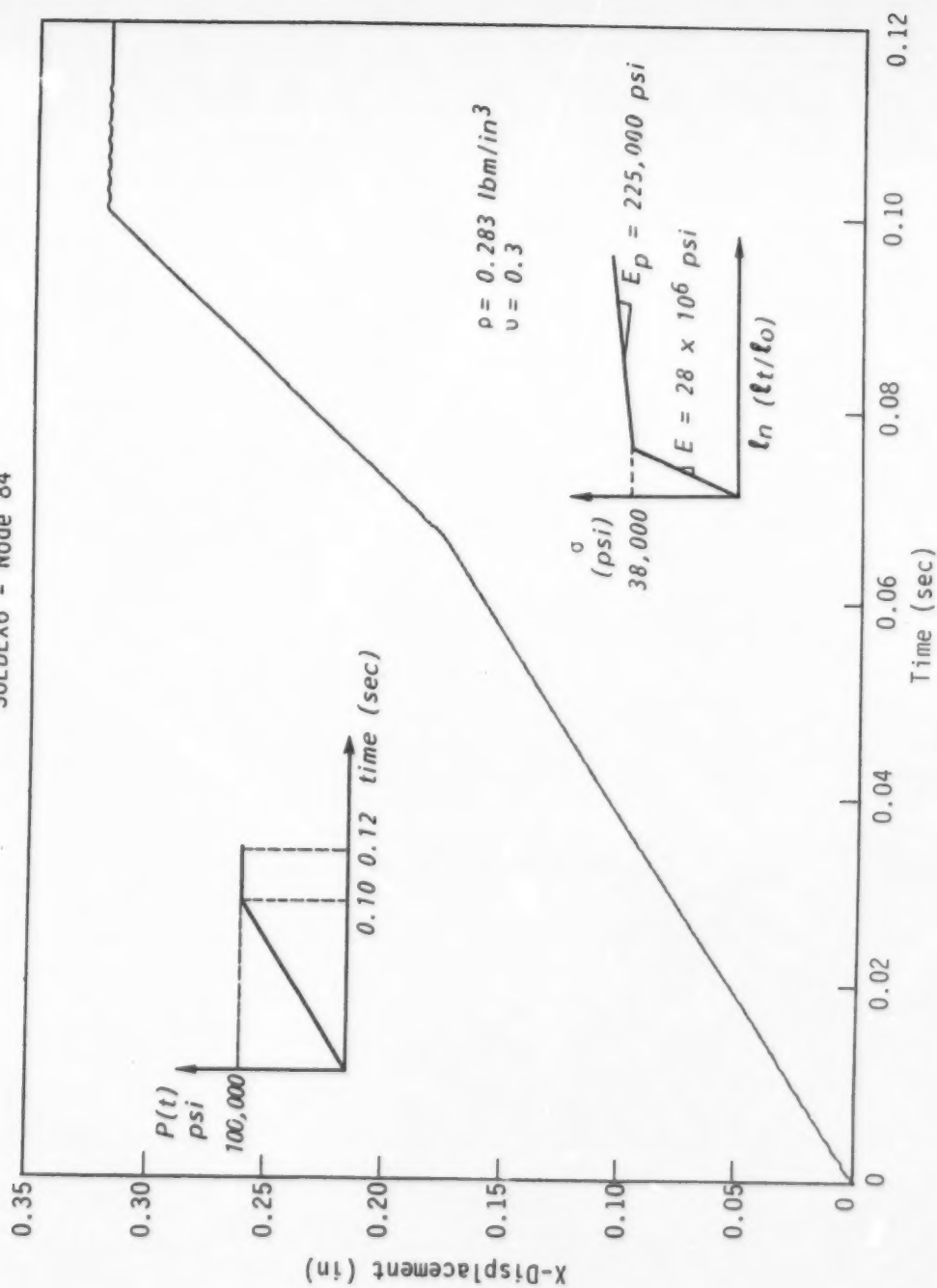


FIGURE 2.55.1
CASE 55: DYNAMIC RELAXATION - ELASTIC-PLASTIC MATERIAL
NODE 84 DISPLACEMENT

SOLDEX6 - Element 20

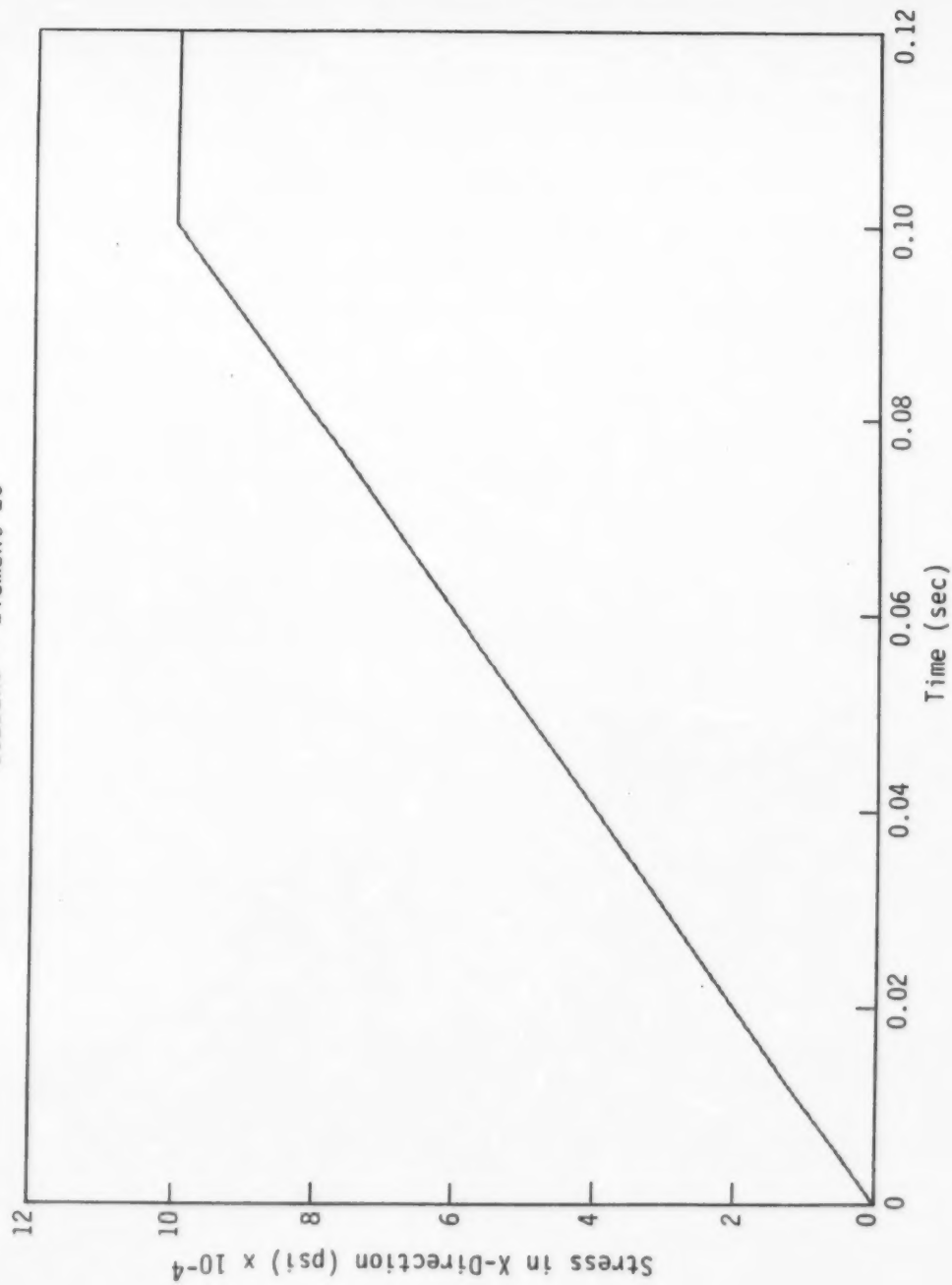


FIGURE 2.55.2

CASE 55: DYNAMIC RELAXATION - ELASTIC-PLASTIC MATERIAL
ELEMENT 20 AXIAL STRESS

SOLDEX6A - Dynamic Relaxation - Node 84

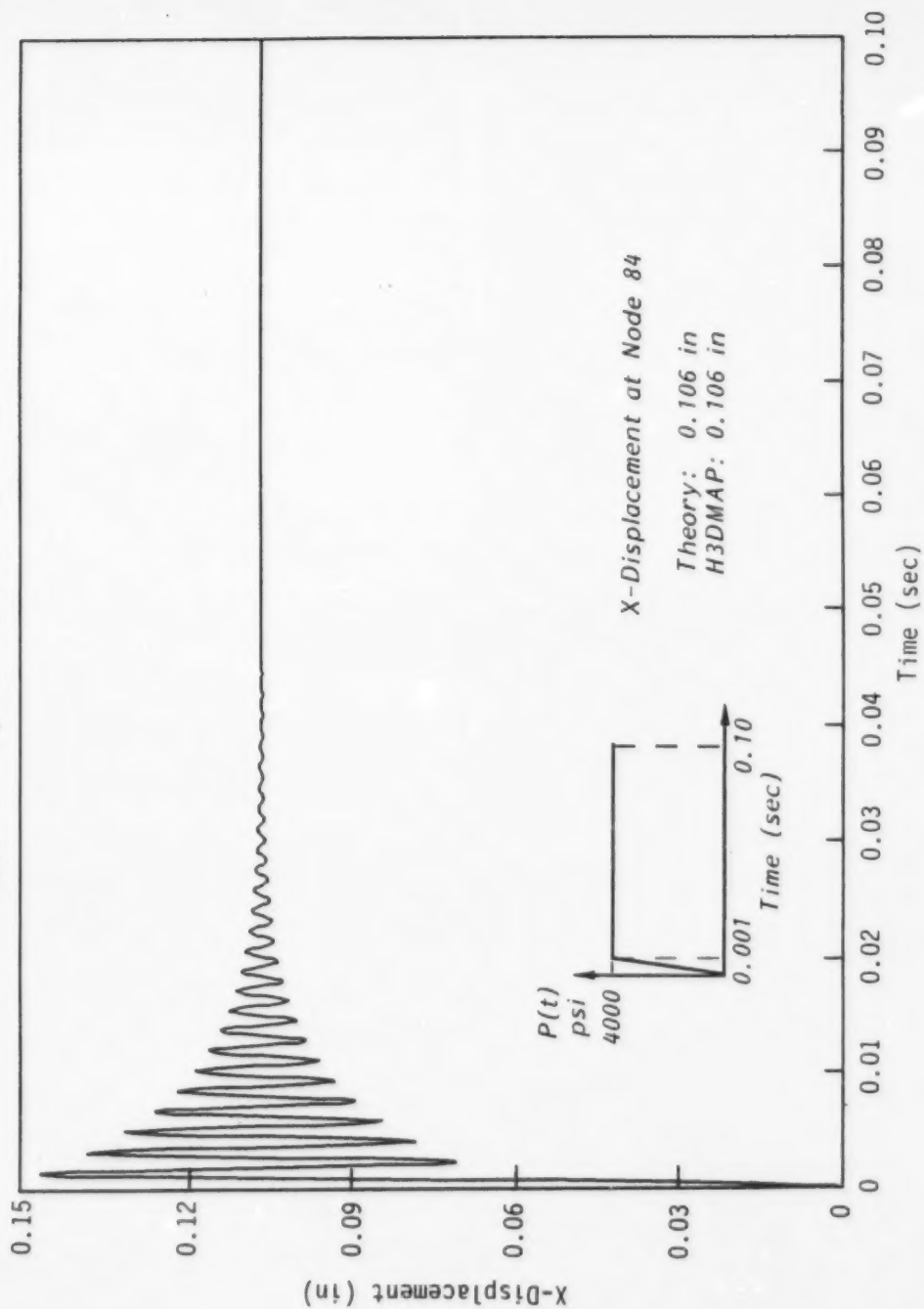


FIGURE 2.55.3
CASE 55: DYNAMIC RELAXATION - ELASTIC MATERIAL
NODE 84 DISPLACEMENT

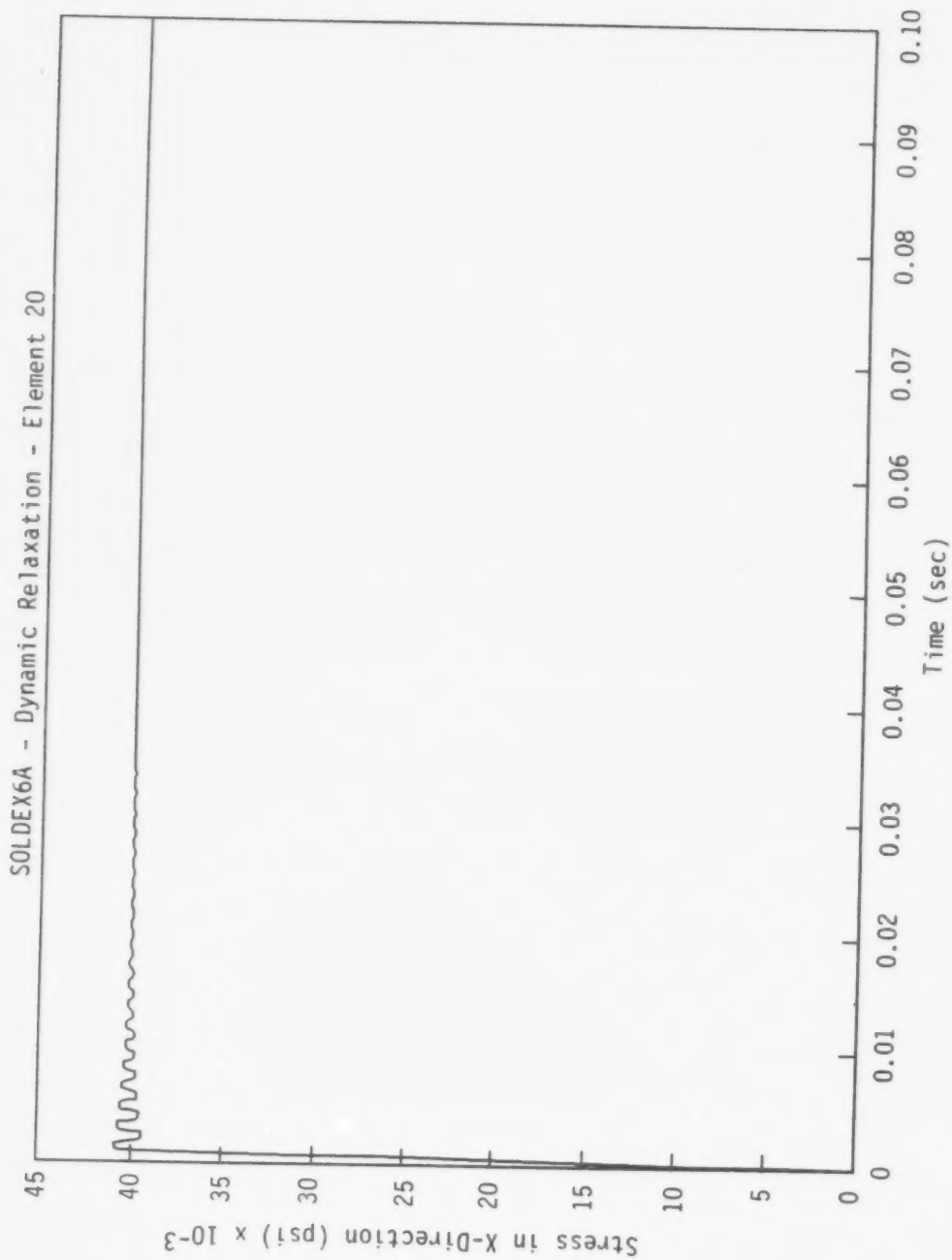


FIGURE 2.55.4
CASE 55: DYNAMIC RELAXATION - ELASTIC MATERIAL
ELEMENT 20 AXIAL STRESS

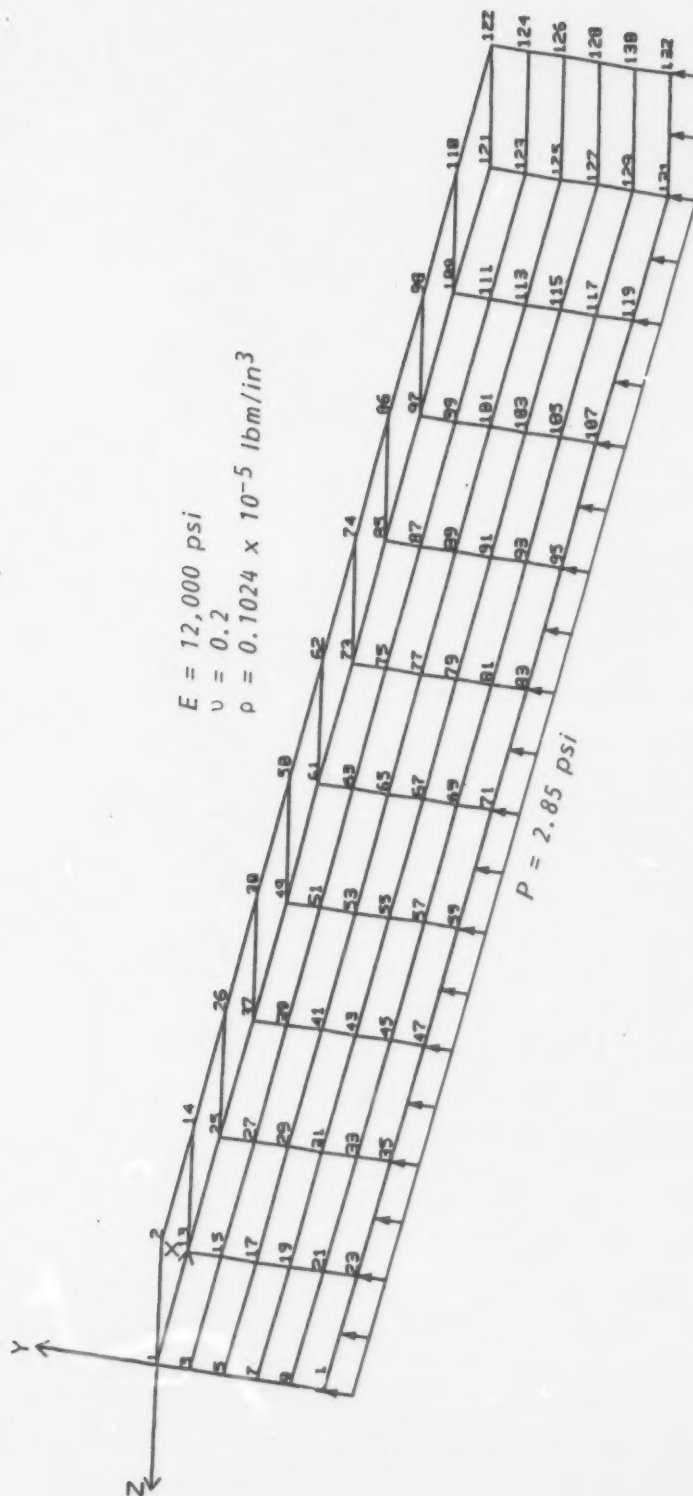
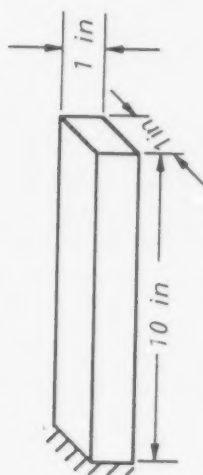


FIGURE 2.56.1
CASE 56: GEOMETRIC NON-LINEAR BEAM TEST MODEL

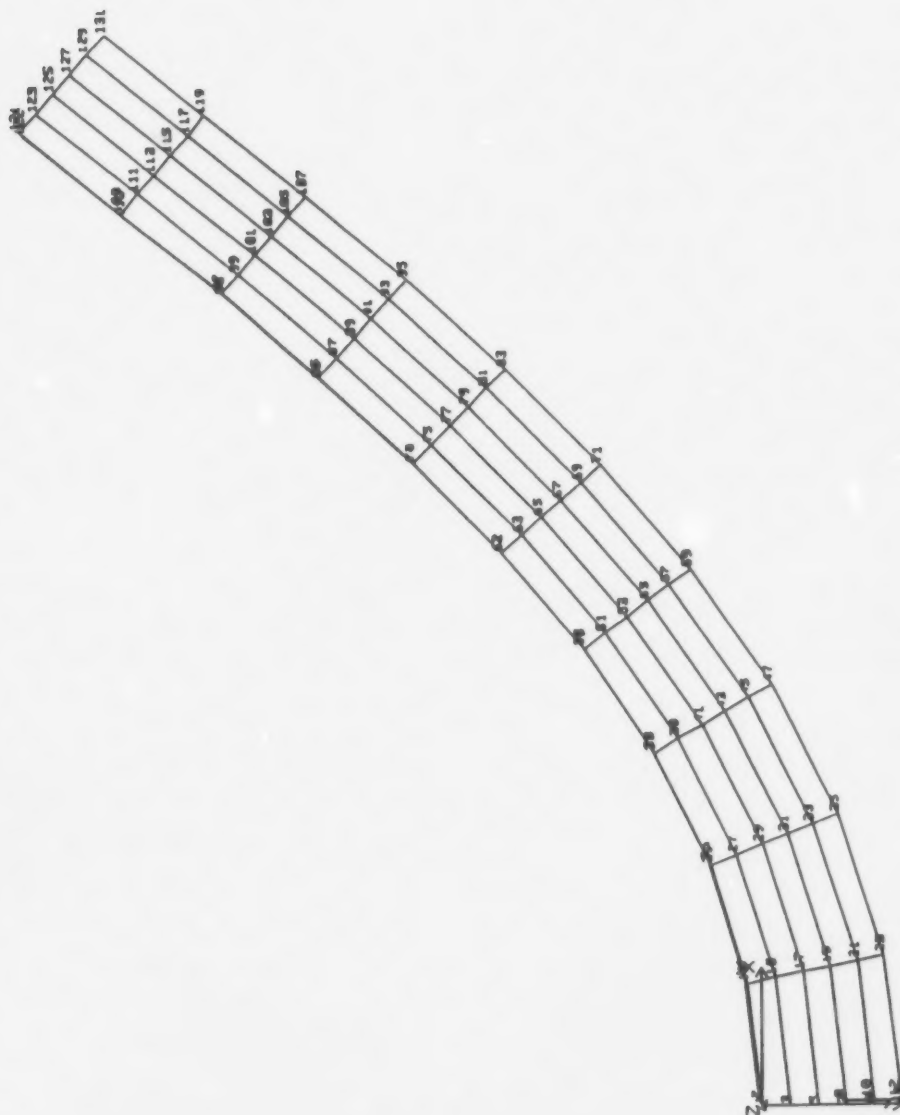


FIGURE 2.56.2
CASE 56: GEOMETRIC NON-LINEAR BEAM TEST
RESPONSE AT TIME: 0.0030 sec

SOLDEX7B - NODE 132

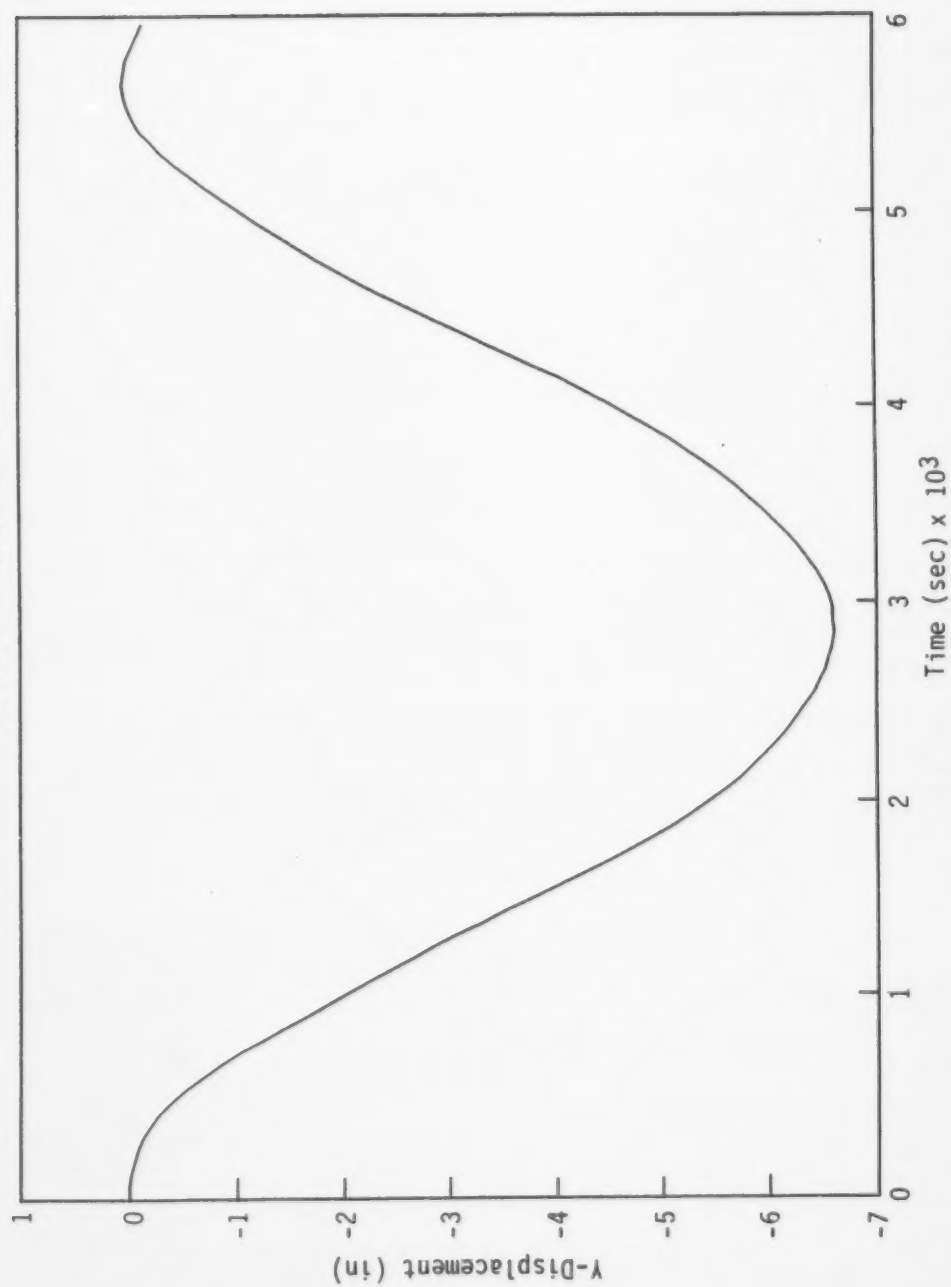


FIGURE 2.56.3
CASE 56: GEOMETRIC NON-LINEAR BEAM TEST
RESPONSE OF NODE 132

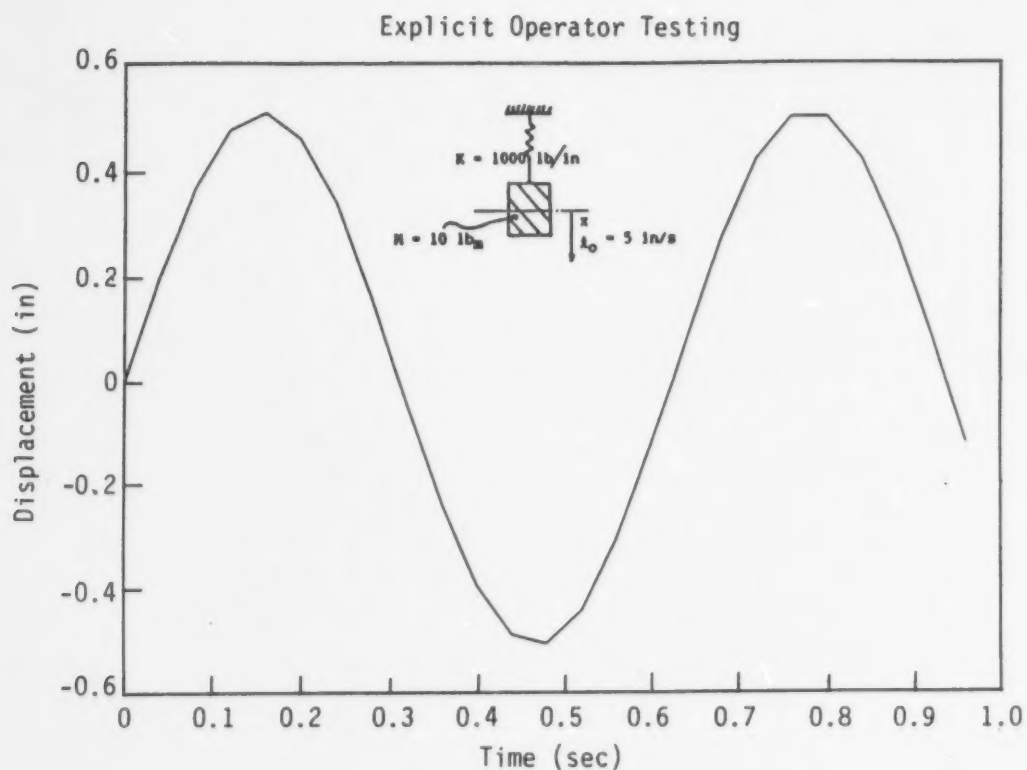


FIGURE 2.57.1

CASE 57: UNDAMPED SPRING-MASS SYSTEM WITH INITIAL VELOCITY
MODEL AND DISPLACEMENT RESPONSE

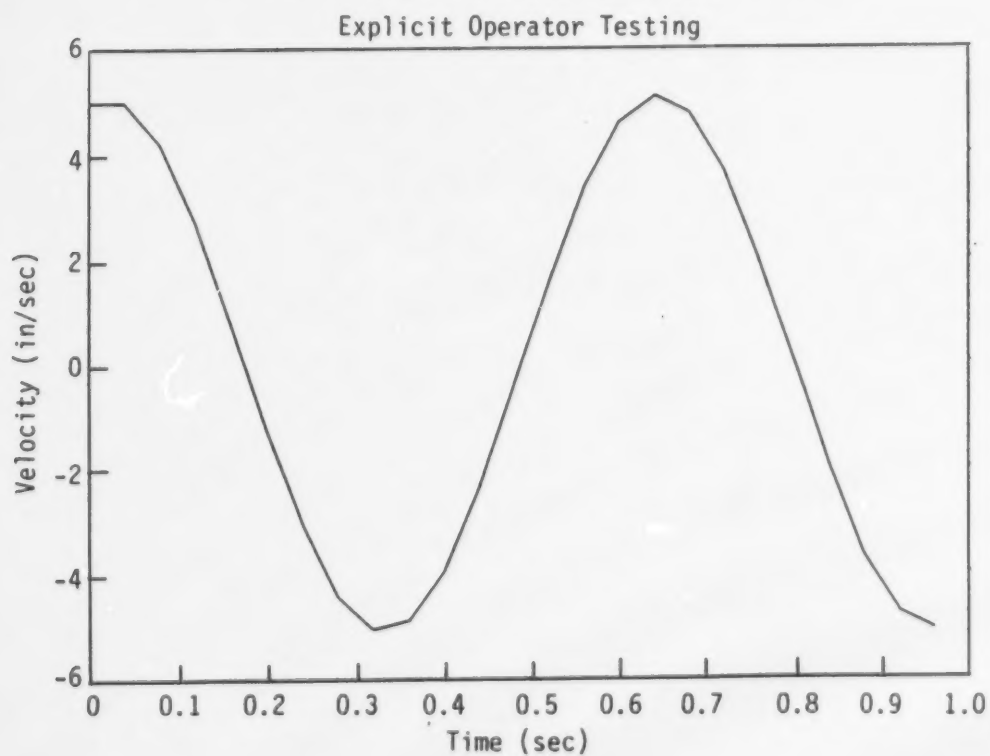


FIGURE 2.57.2

CASE 57: UNDAMPED SPRING-MASS SYSTEM WITH INITIAL VELOCITY
VELOCITY RESPONSE

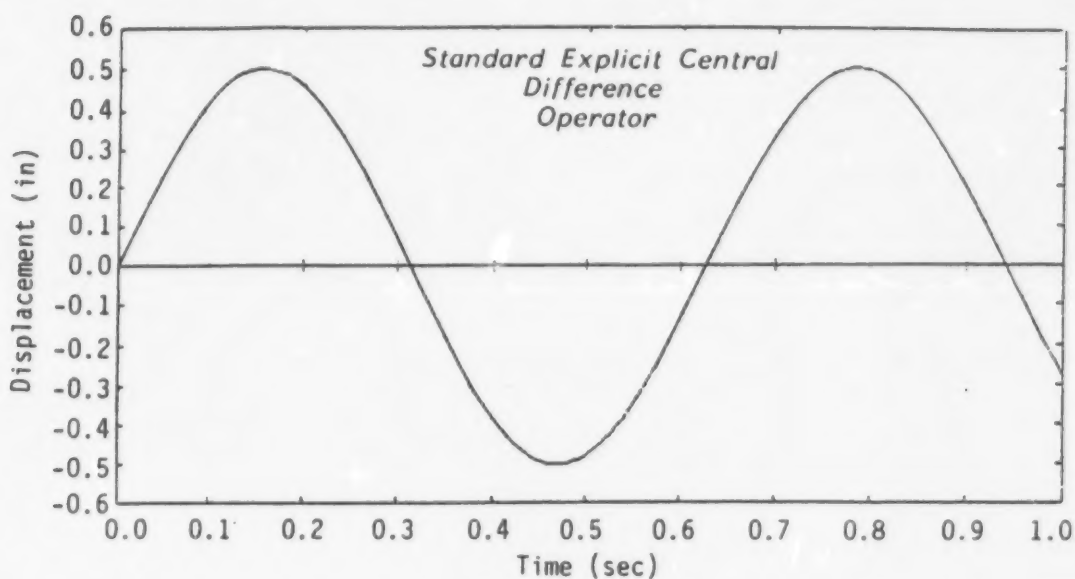


FIGURE 2.57.3
CASE 57: THEORETICAL SOLUTION
DISPLACEMENT vs TIME

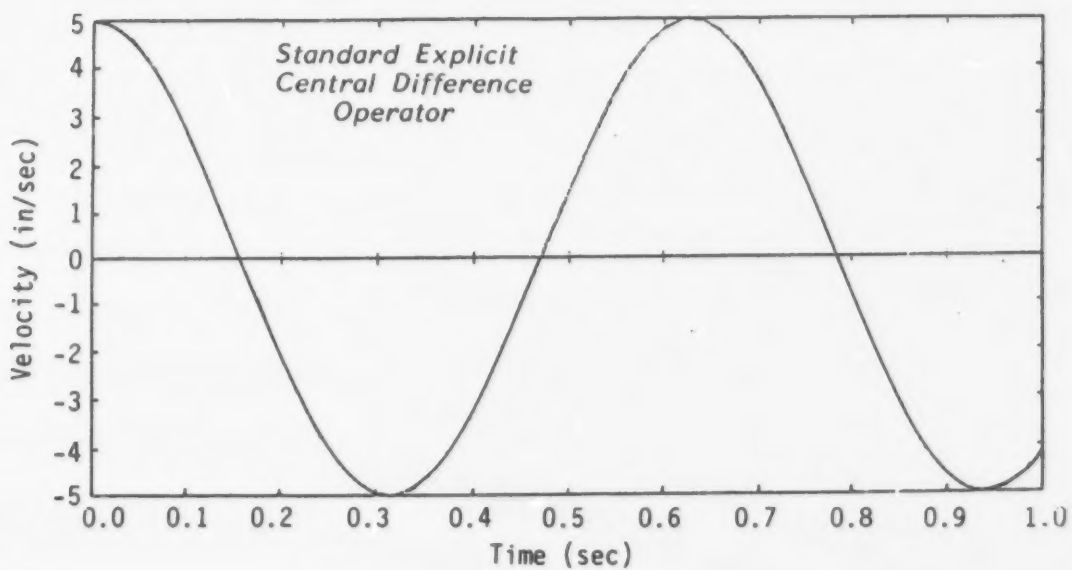
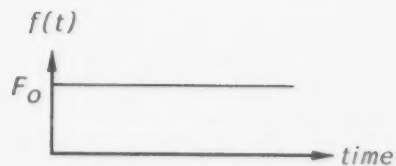
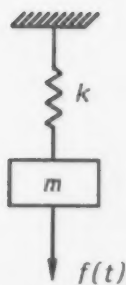
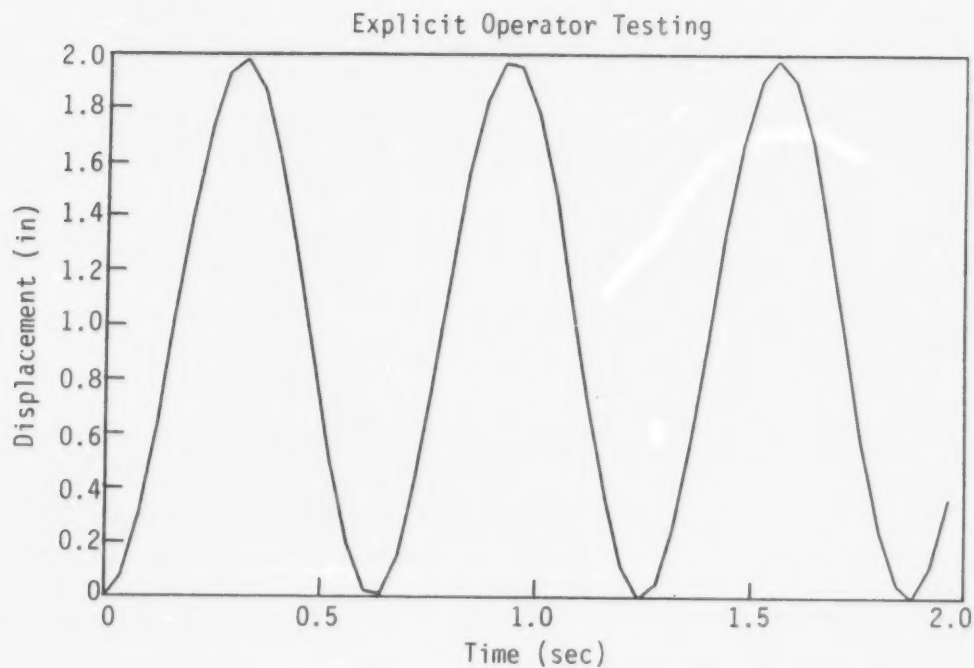


FIGURE 2.57.4
CASE 57: THEORETICAL SOLUTION
VELOCITY vs TIME



$$\begin{aligned}
 m &= 10 \text{ lb sec}^2/\text{in} \\
 k &= 1000 \text{ lb/in} \\
 F_0 &= 990 \text{ lbs}
 \end{aligned}$$

Theoretical Results

$$\begin{aligned}
 \text{Natural frequency: } &10 \text{ rad/sec} \\
 \text{Period: } &0.628 \text{ sec} \\
 \text{Amplitude: } &0.99 \text{ in (static)}
 \end{aligned}$$

FIGURE 2.58.1
CASE 58: UNDAMPED SPRING-MASS SYSTEM
WITH FORCE STEP FUNCTION

Explicit Operator Testing

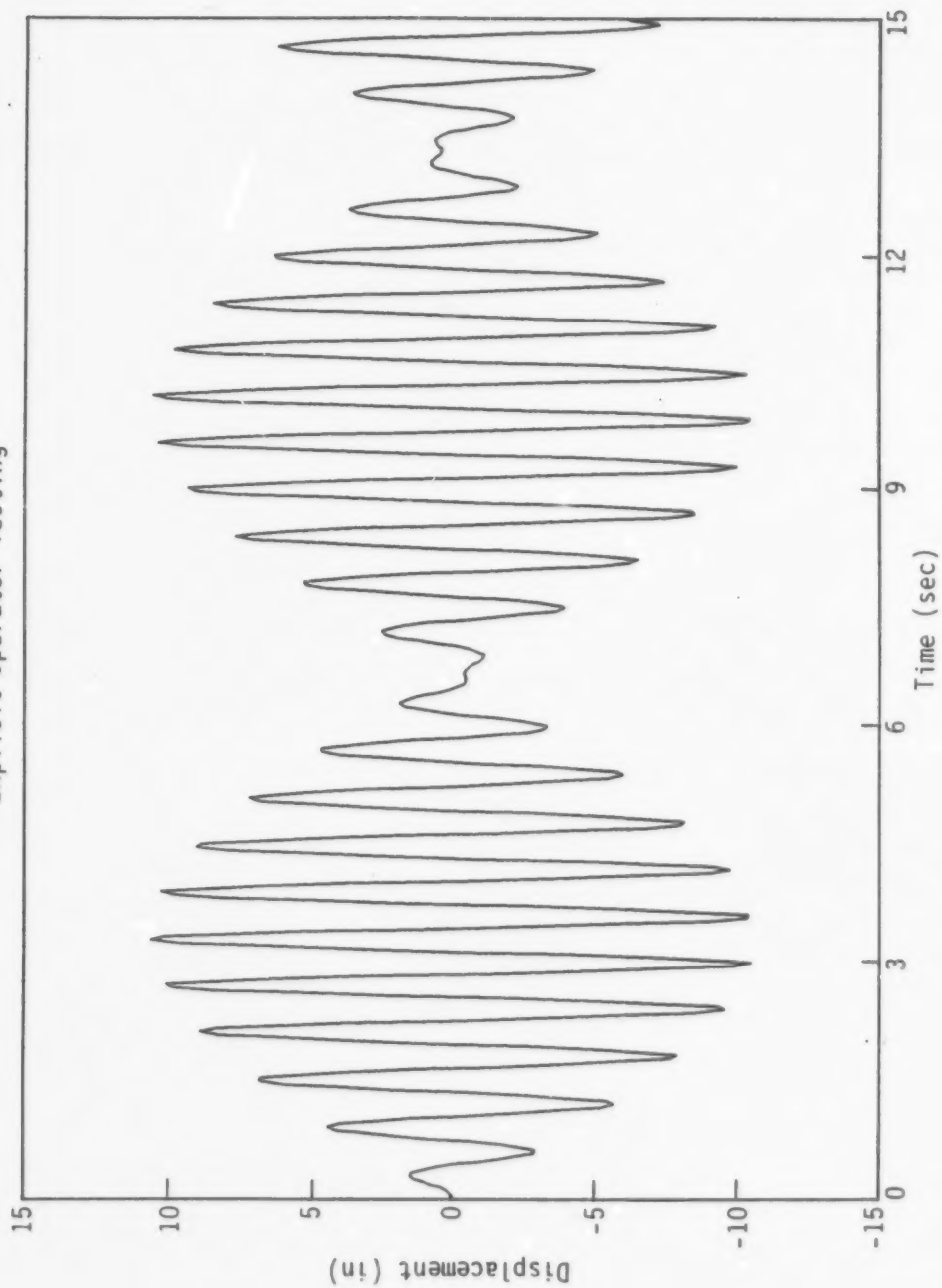


FIGURE 2.59.1
CASE 59: UNDAMPED SPRING-MASS SYSTEM
BEAT RESPONSE

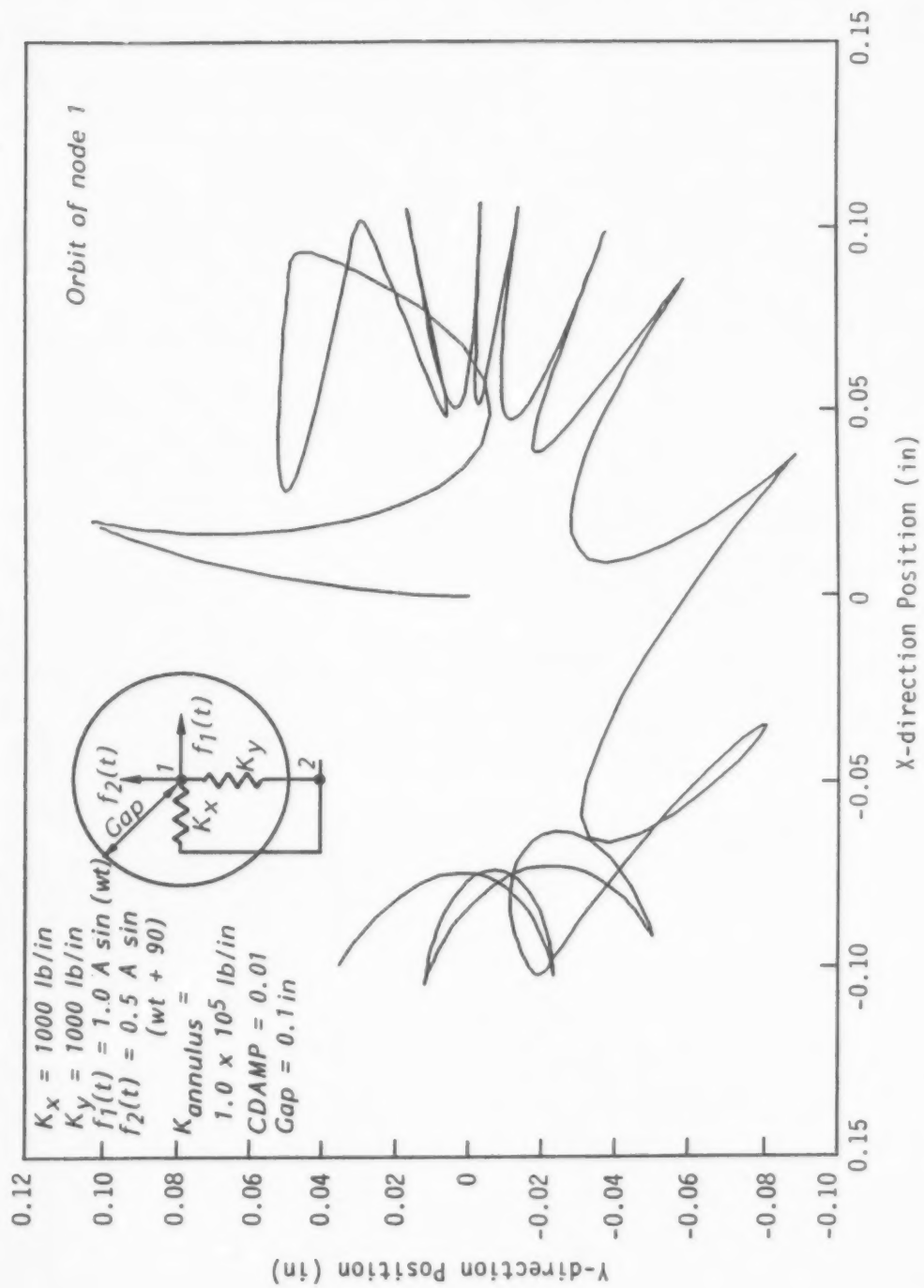
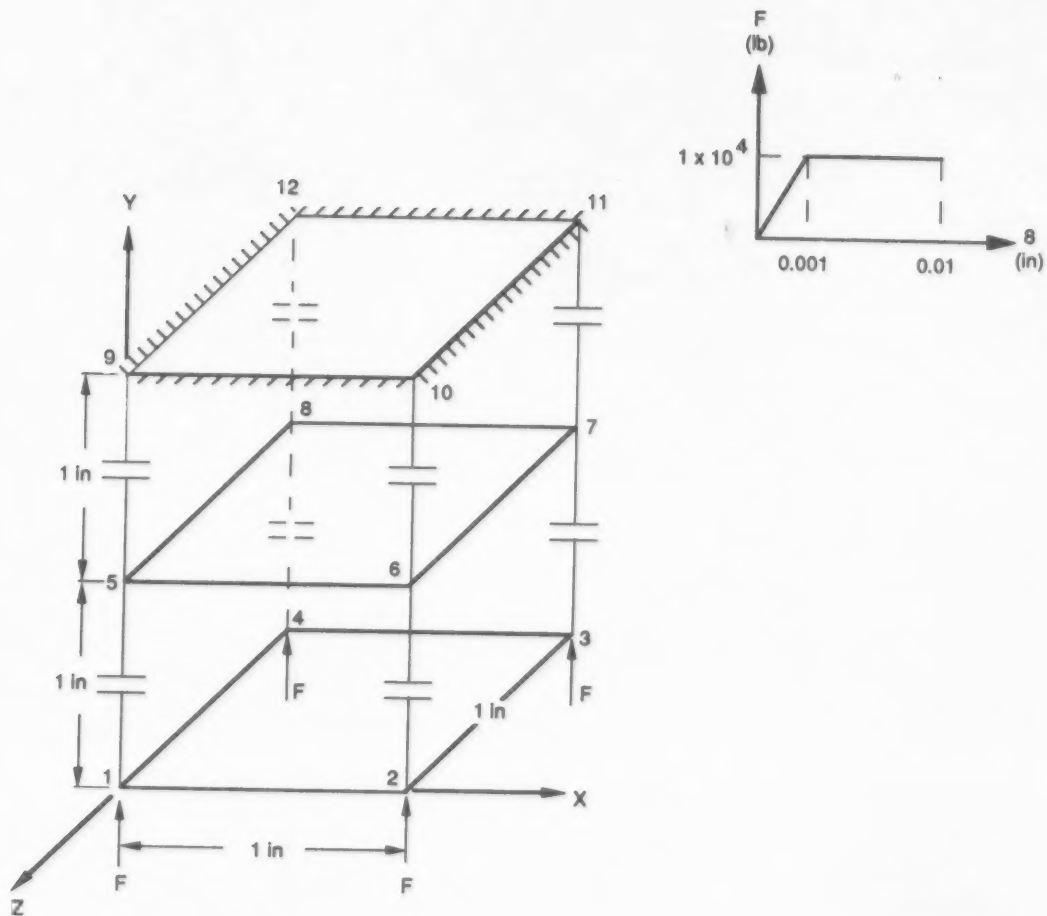


FIGURE 2.60.1
CASE 60: ANNULAR ELEMENT TESTING (EXPLICIT SOLUTION)



X Gap Test 1: Gap Elements
X Gap Test 1A: Spring Elements

$k = 1.0 \times 10^5$ lb/in for both spring or gap elements

Results	Theory	H3DMAP
Deflection at Nodes 1-4	0.2 in	0.2 in
Deflection at Nodes 5-8	0.1 in	0.1 in

FIGURE 2.61.1

CASE 61: SANDWICH MODEL

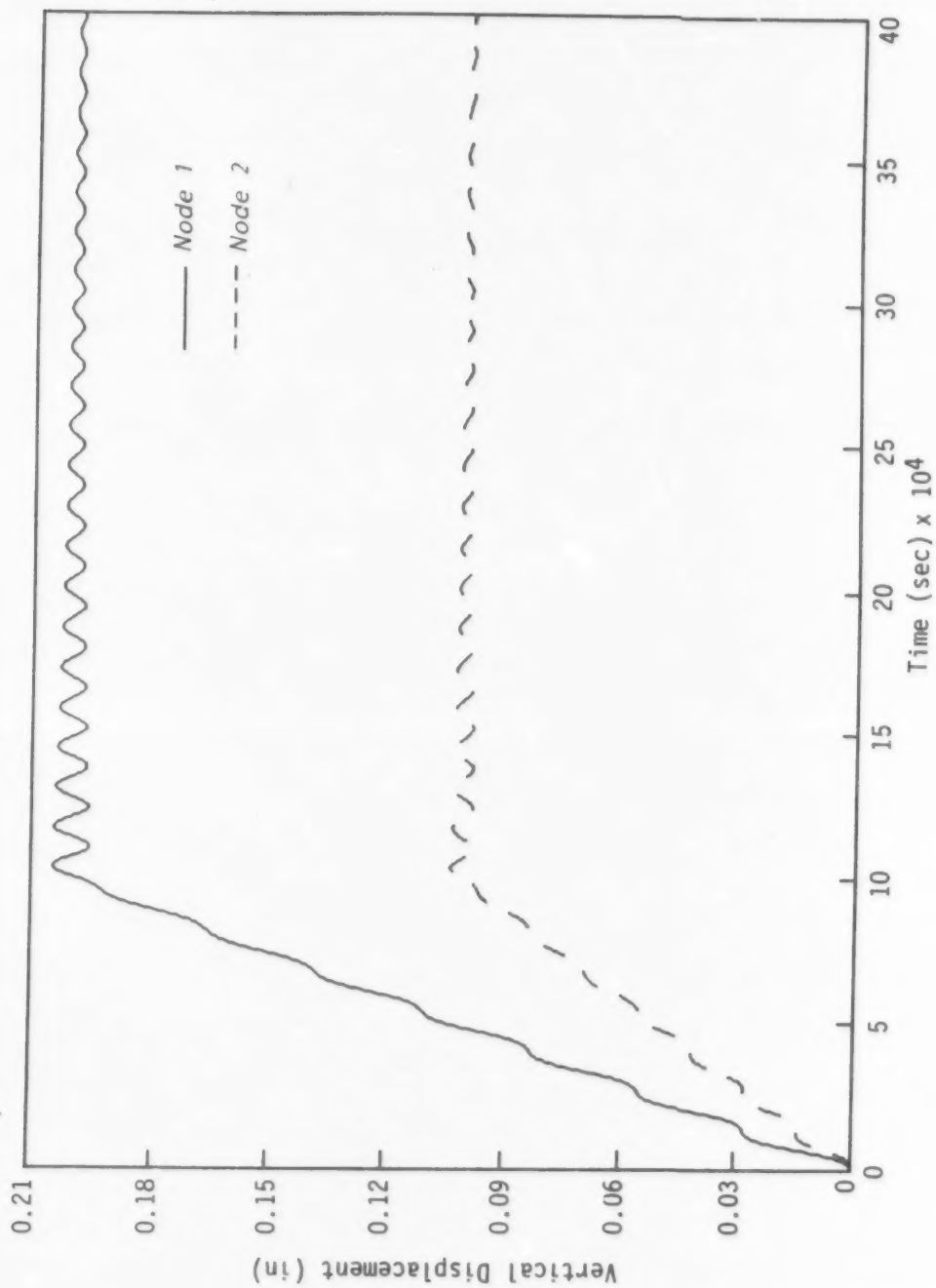


FIGURE 2.61.2
CASE 61: SANDWICH MODEL
DISPLACEMENT RESPONSE

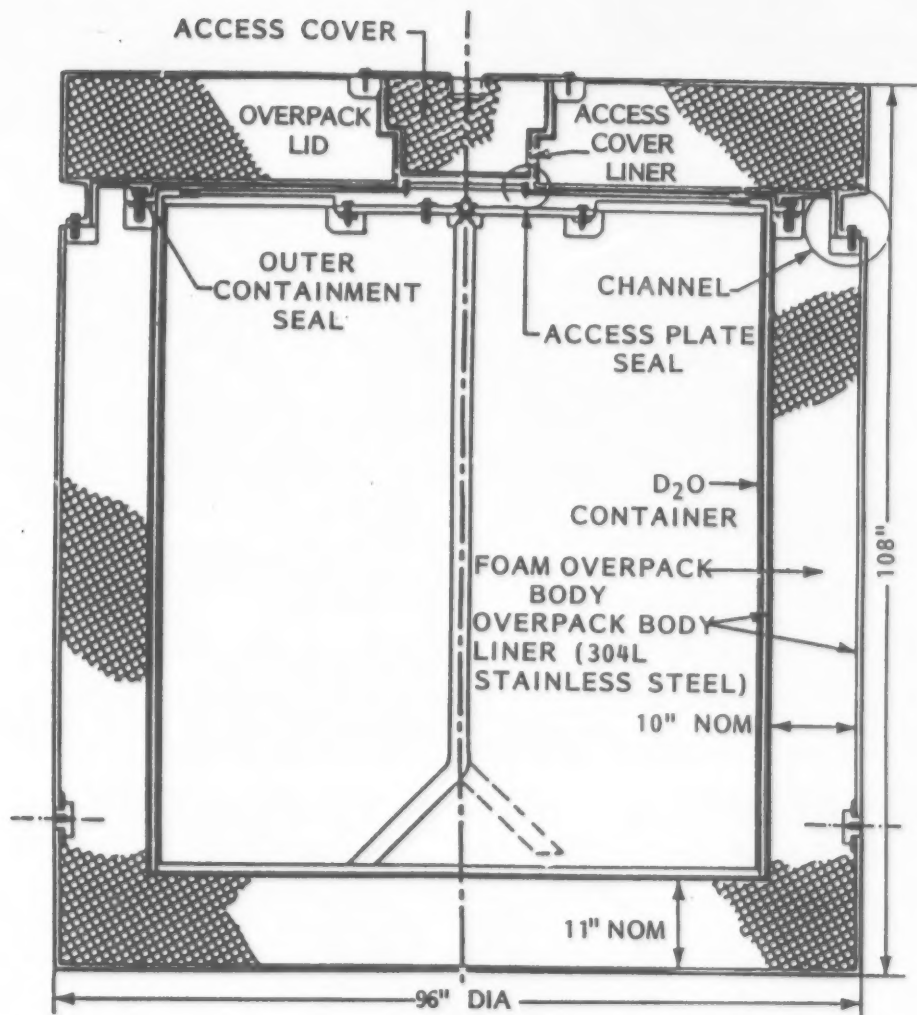


FIGURE 2.62.1
CASE 62: TRITIATED HEAVY WATER TRANSPORTATION PACKAGE

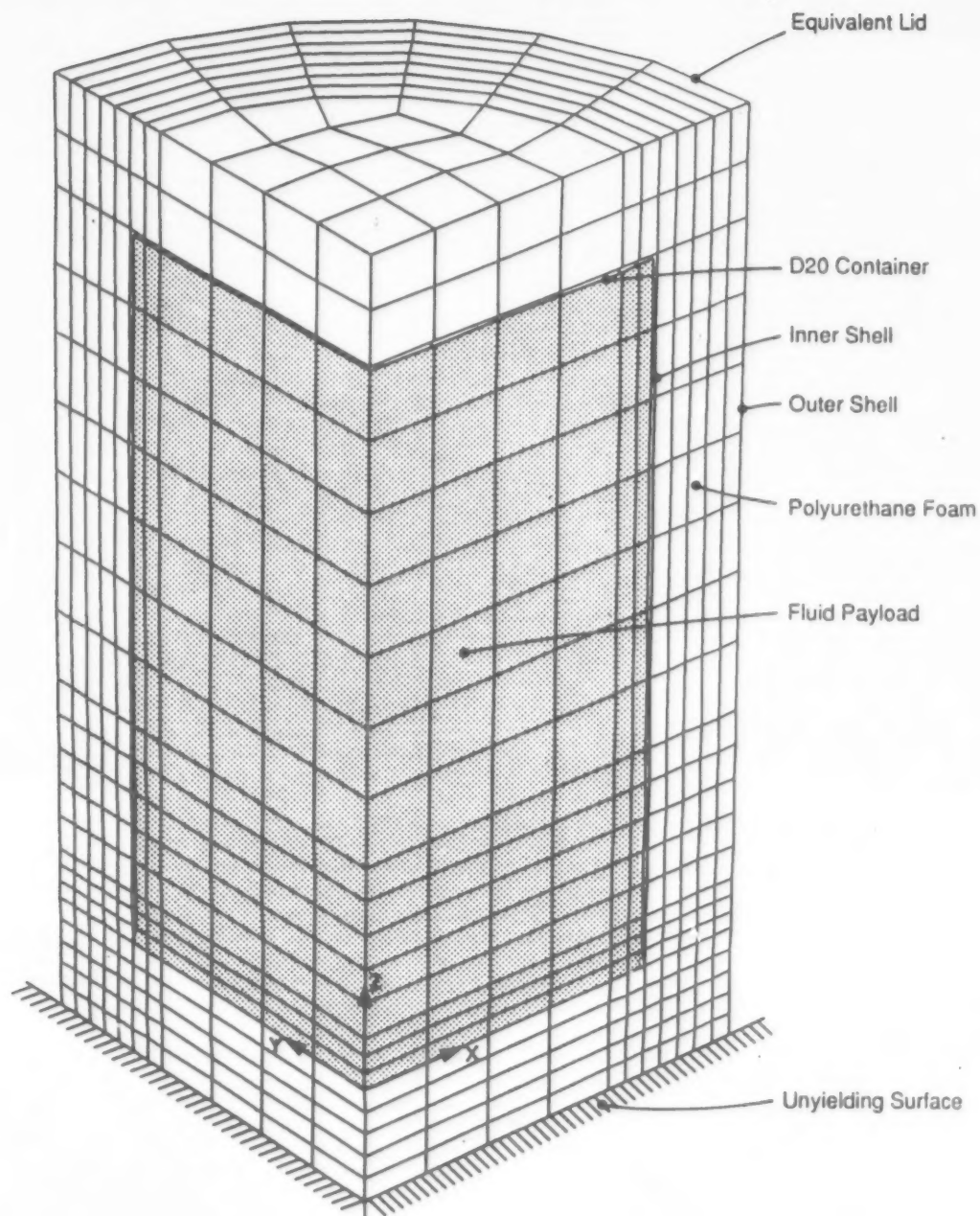


FIGURE 2.62.2
CASE 62: OVERALL FULL SCALE THWTP FSI MODEL GEOMETRY

182257-G20-RD

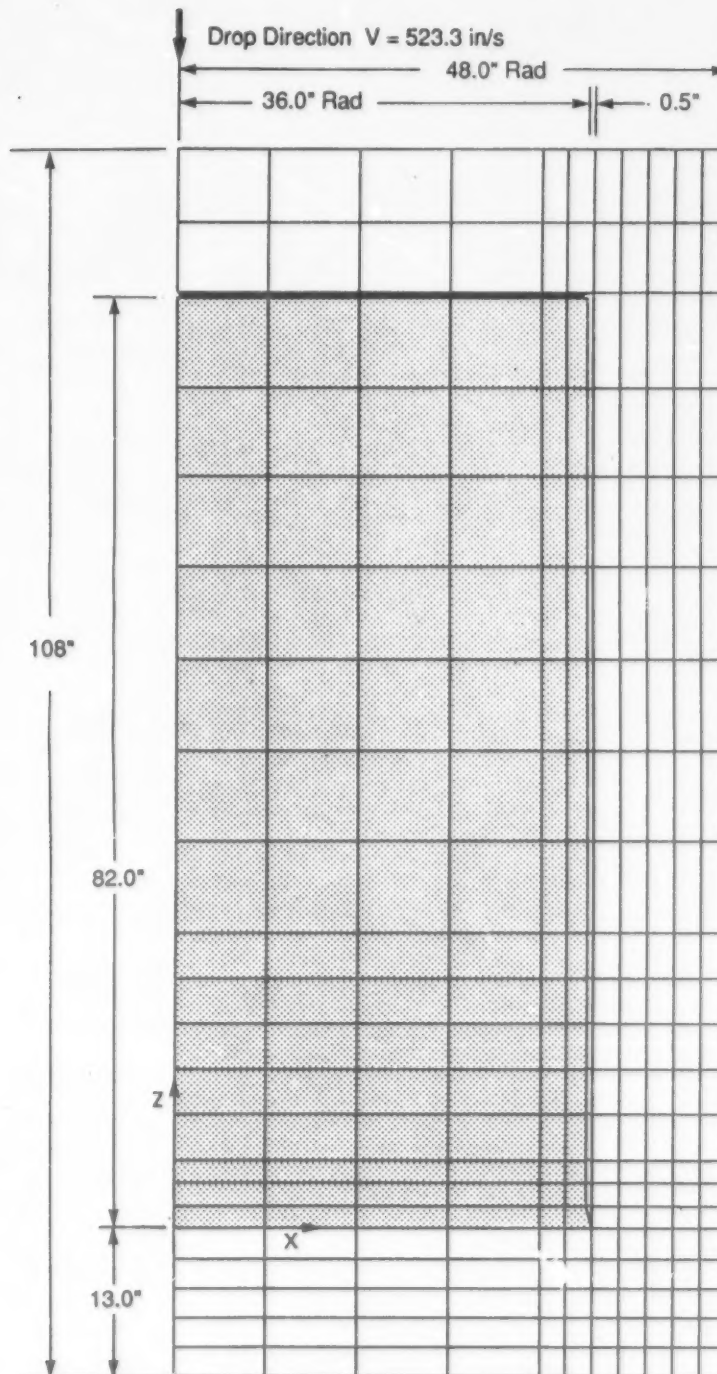
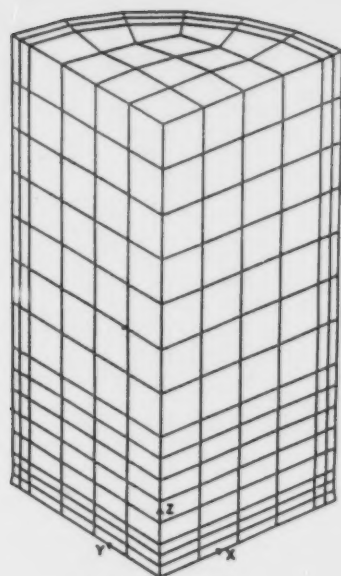


FIGURE 2.62.3
CASE 62: THWTP FSI MODEL DIMENSIONS

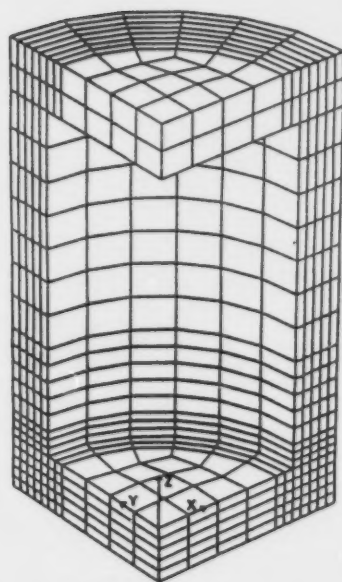
182258-G20-RD



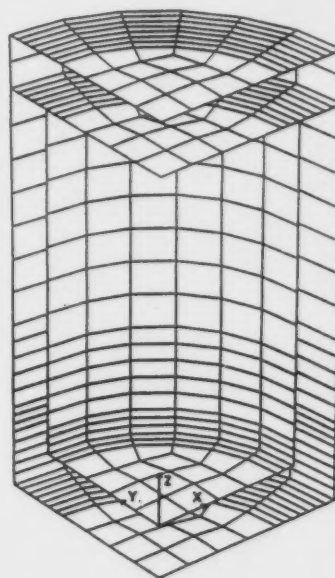
Fluid Elements



Fluid Container Elements



Polyurethane Elements



Package Shell Elements

FIGURE 2.62.3 a
CASE 62: EXPLODED VIEW OF THE MODEL

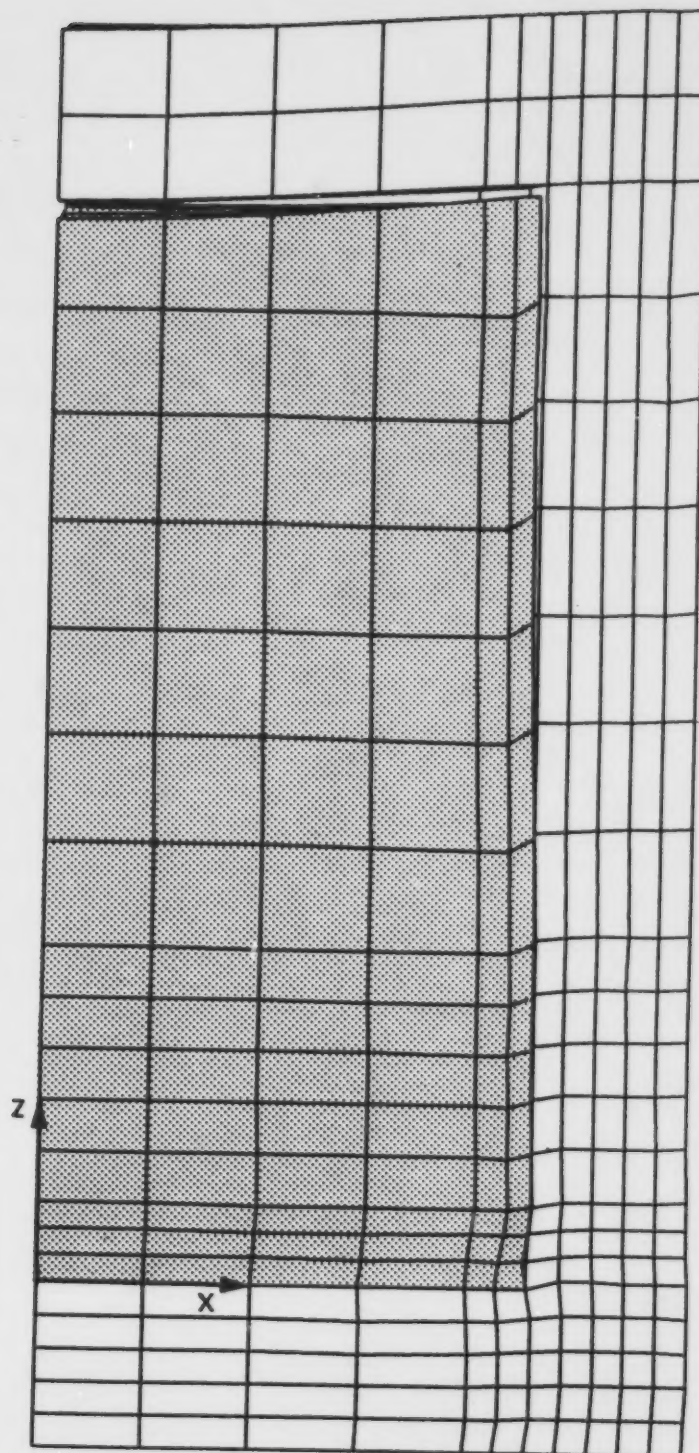


FIGURE 2.62.4
CASE 62: DEFORMED VIEW ON PLANE ($X = 0.0$)
TIME = 0.0050 SEC.

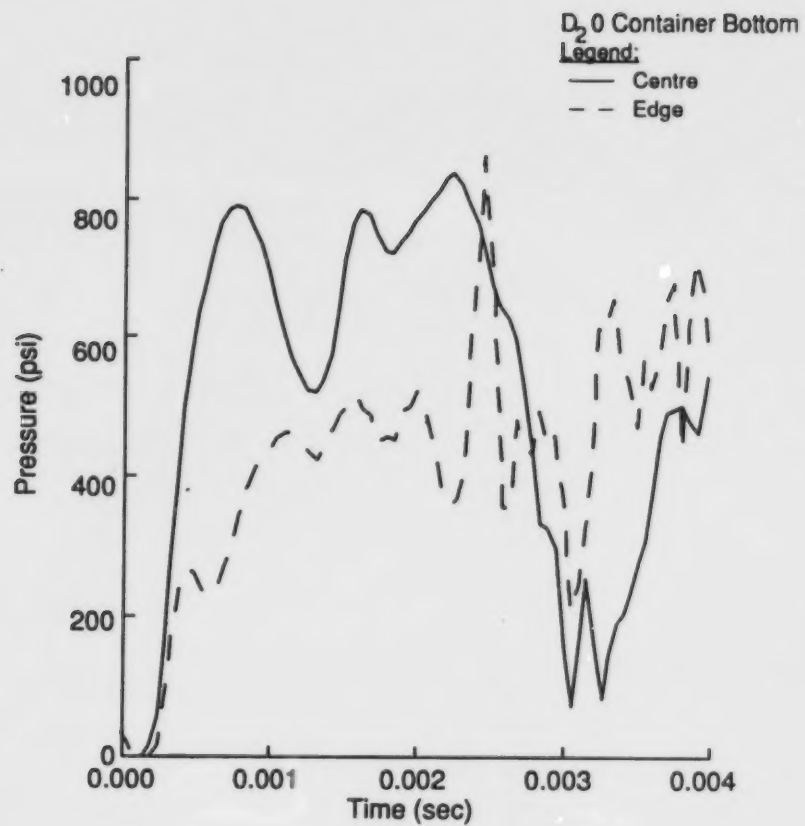


FIGURE 2.62.5
CASE 62: ANALYTICAL RESULTS (FULL SCALE MODEL)

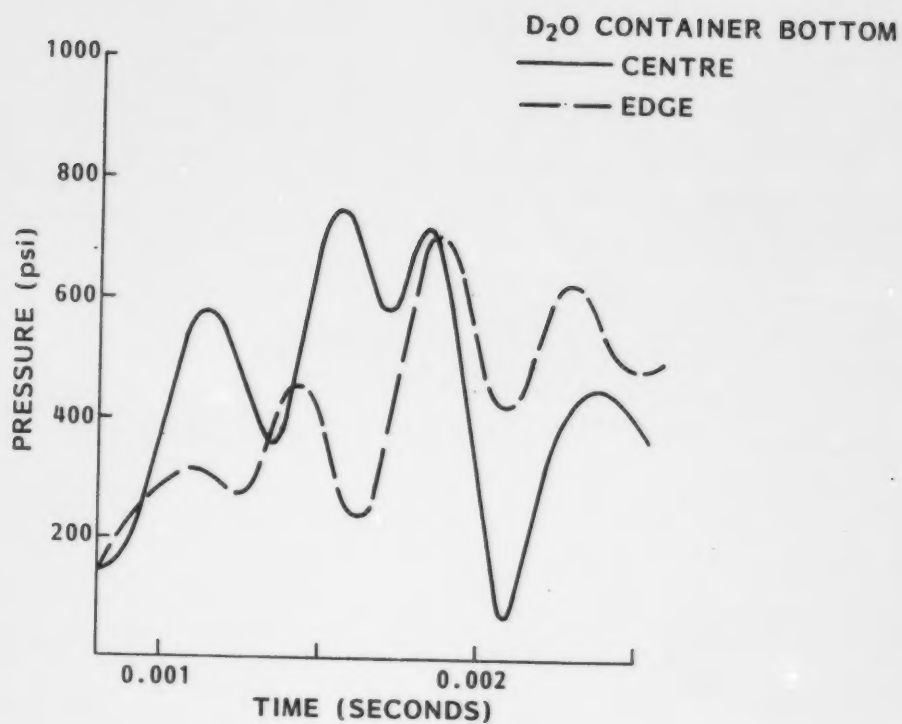
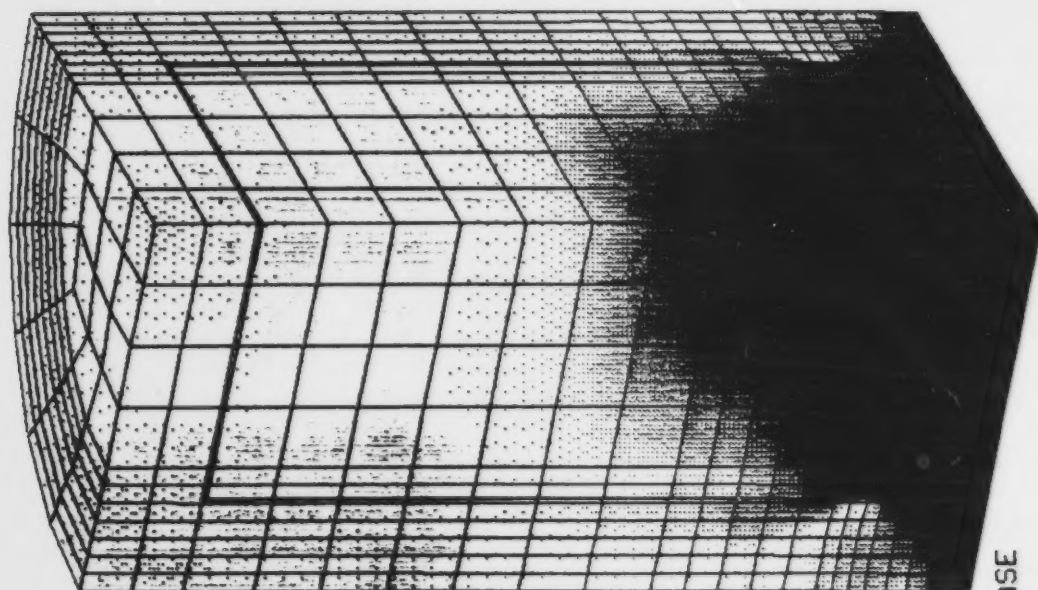
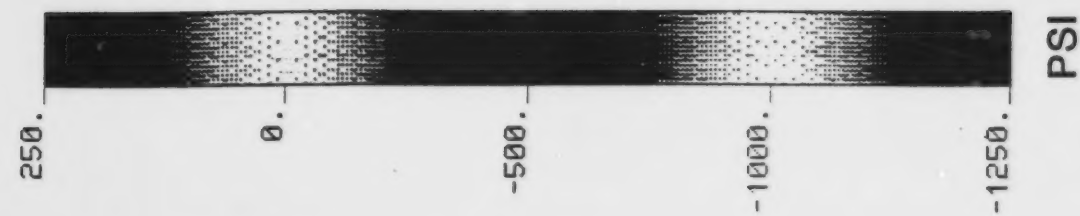


FIGURE 2.62.6
CASE 62: TEST RESULTS (1/4 SCALE MODEL)



THWJTPF VERIFICATION CASE

FIGURE 2.62.7
CASE 62: HYDRODYNAMIC STRESS DISTRIBUTION -
TIME = 1 ms

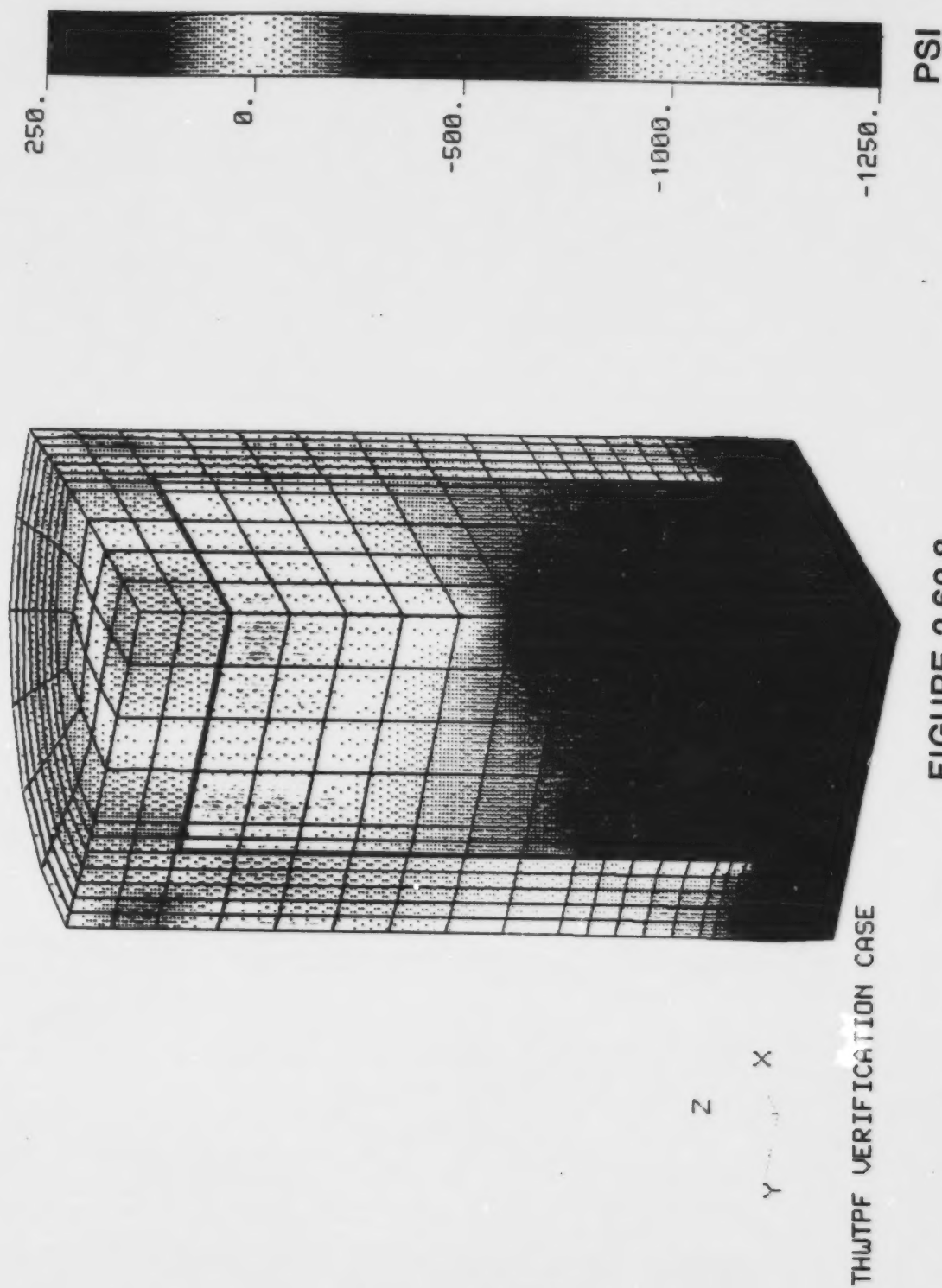
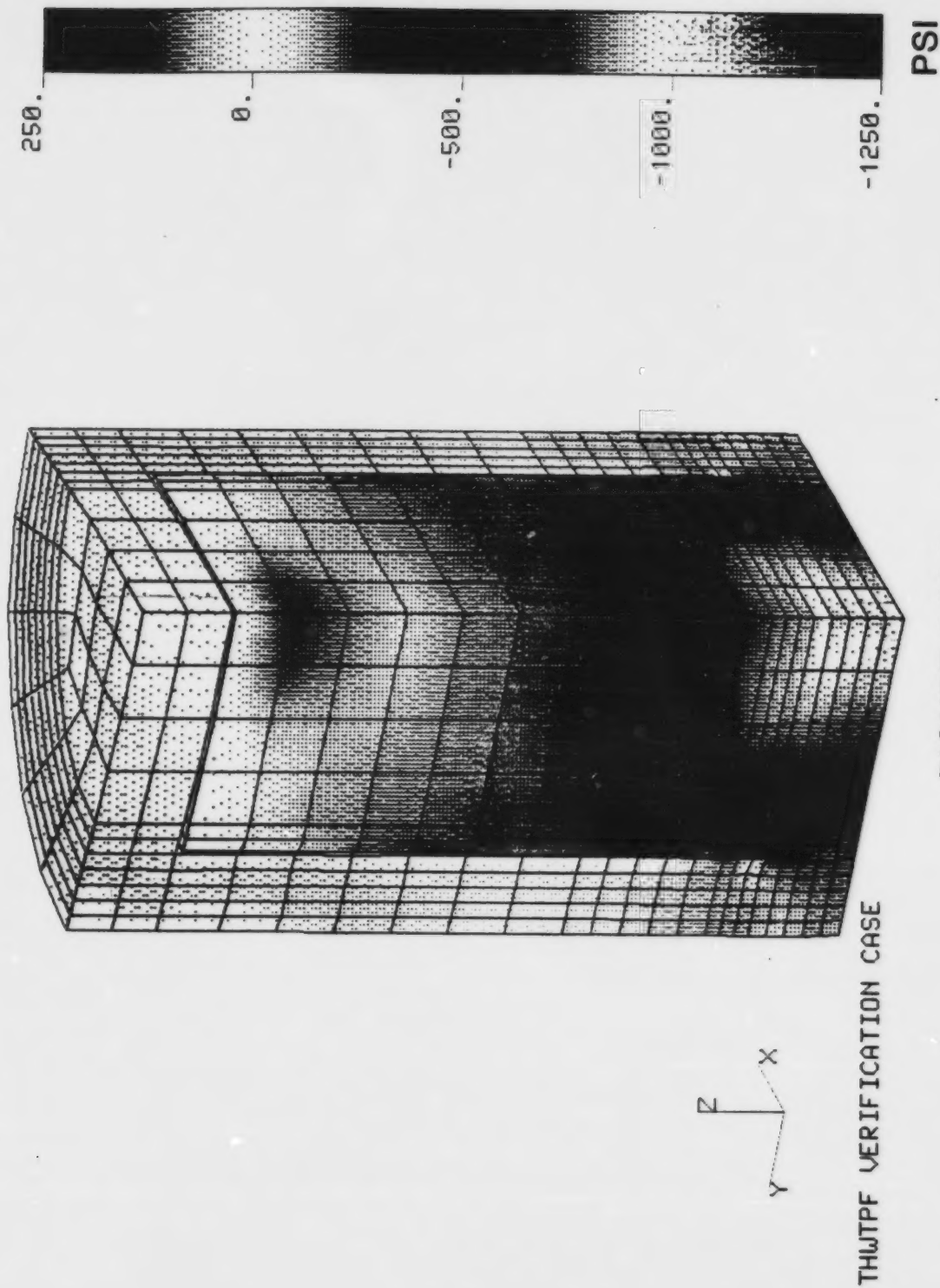


FIGURE 2.62.8
CASE 62: HYDRODYNAMIC STRESS DISTRIBUTION -
TIME = 2 ms



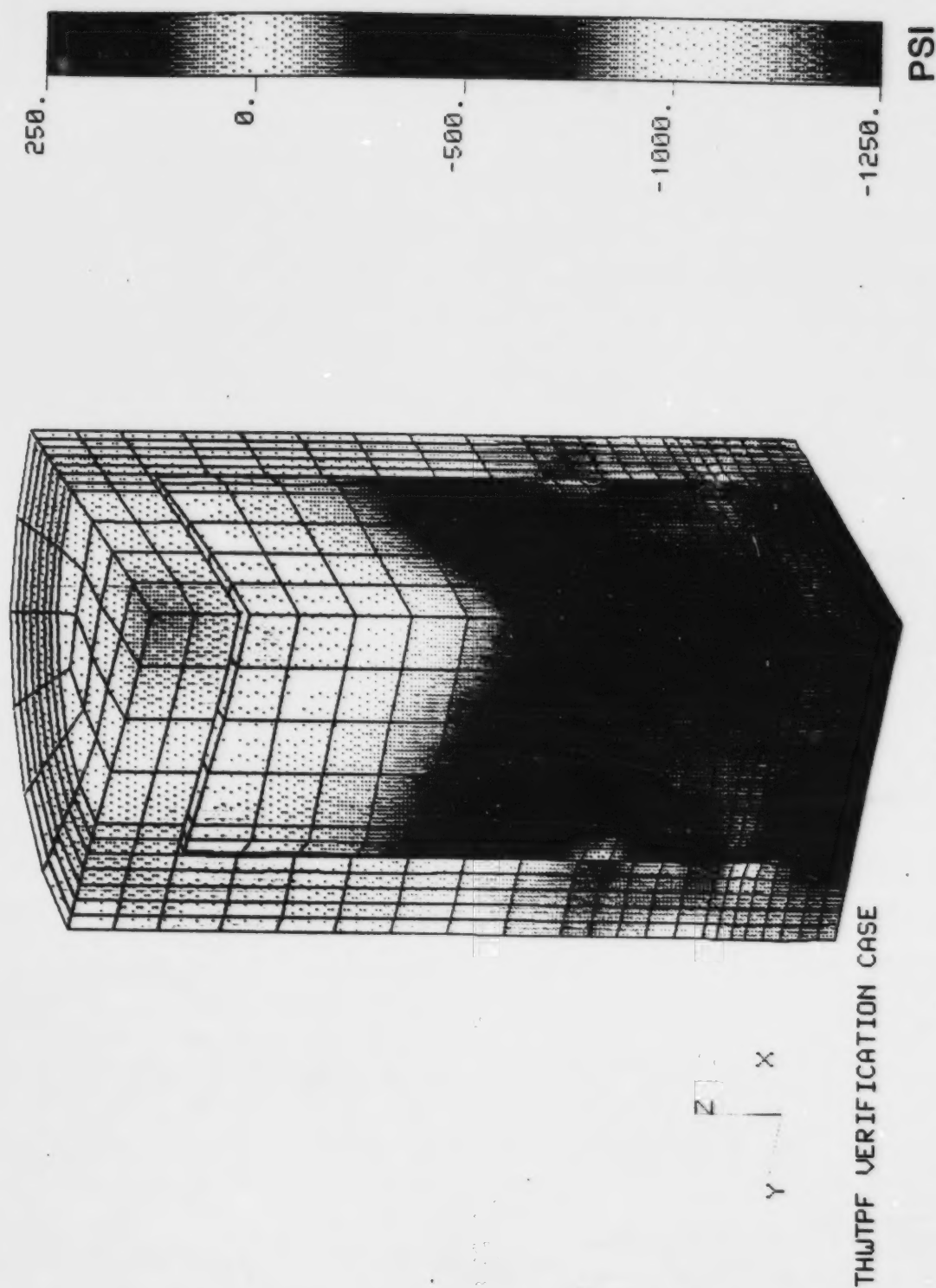


FIGURE 2.62.10
CASE 66: HYDRODYNAMIC STRESS DISTRIBUTION -
TIME = 5 ms

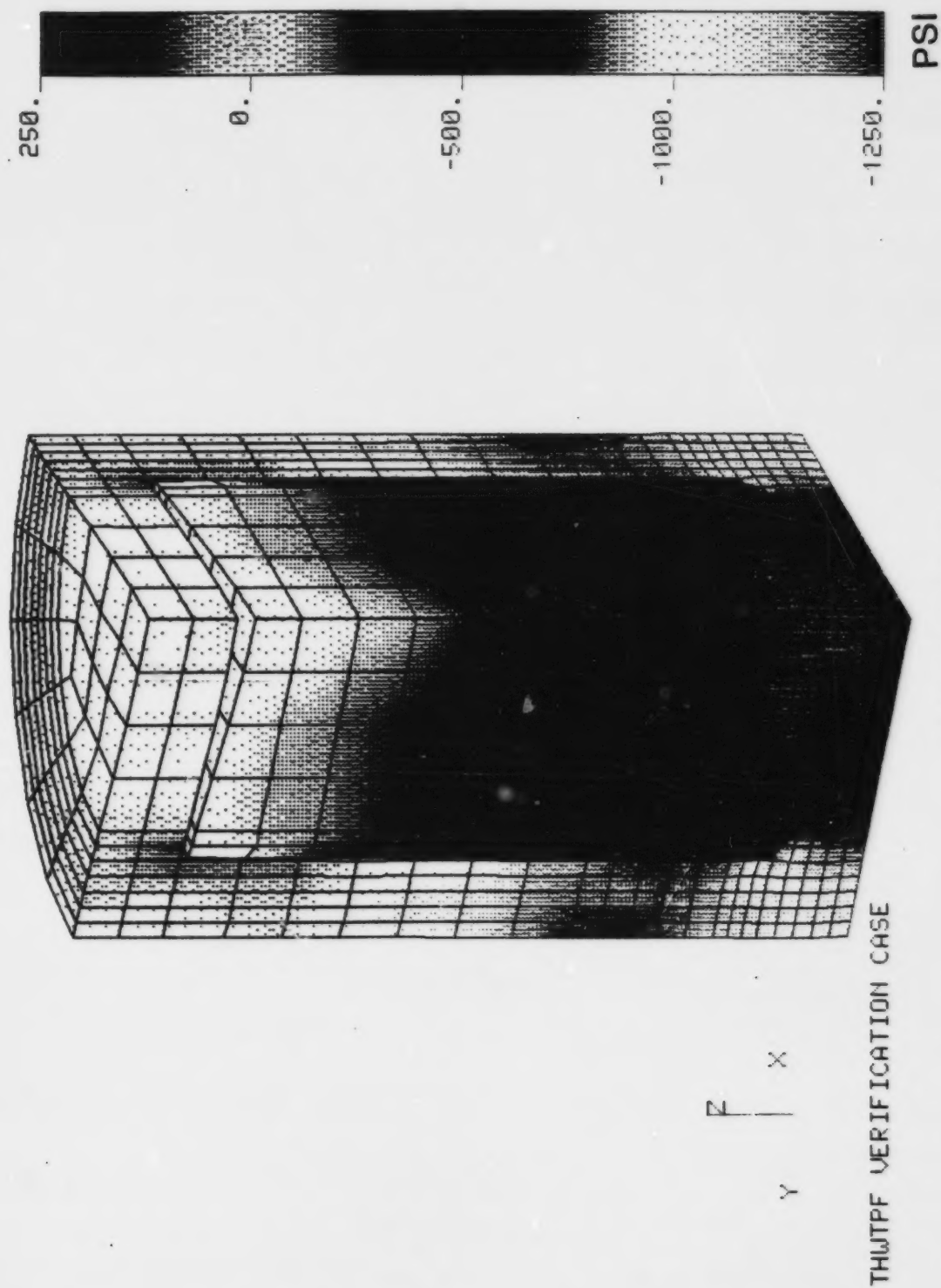


FIGURE 2.62.11
CASE 67: HYDRODYNAMIC STRESS DISTRIBUTION -
TIME = 6 ms

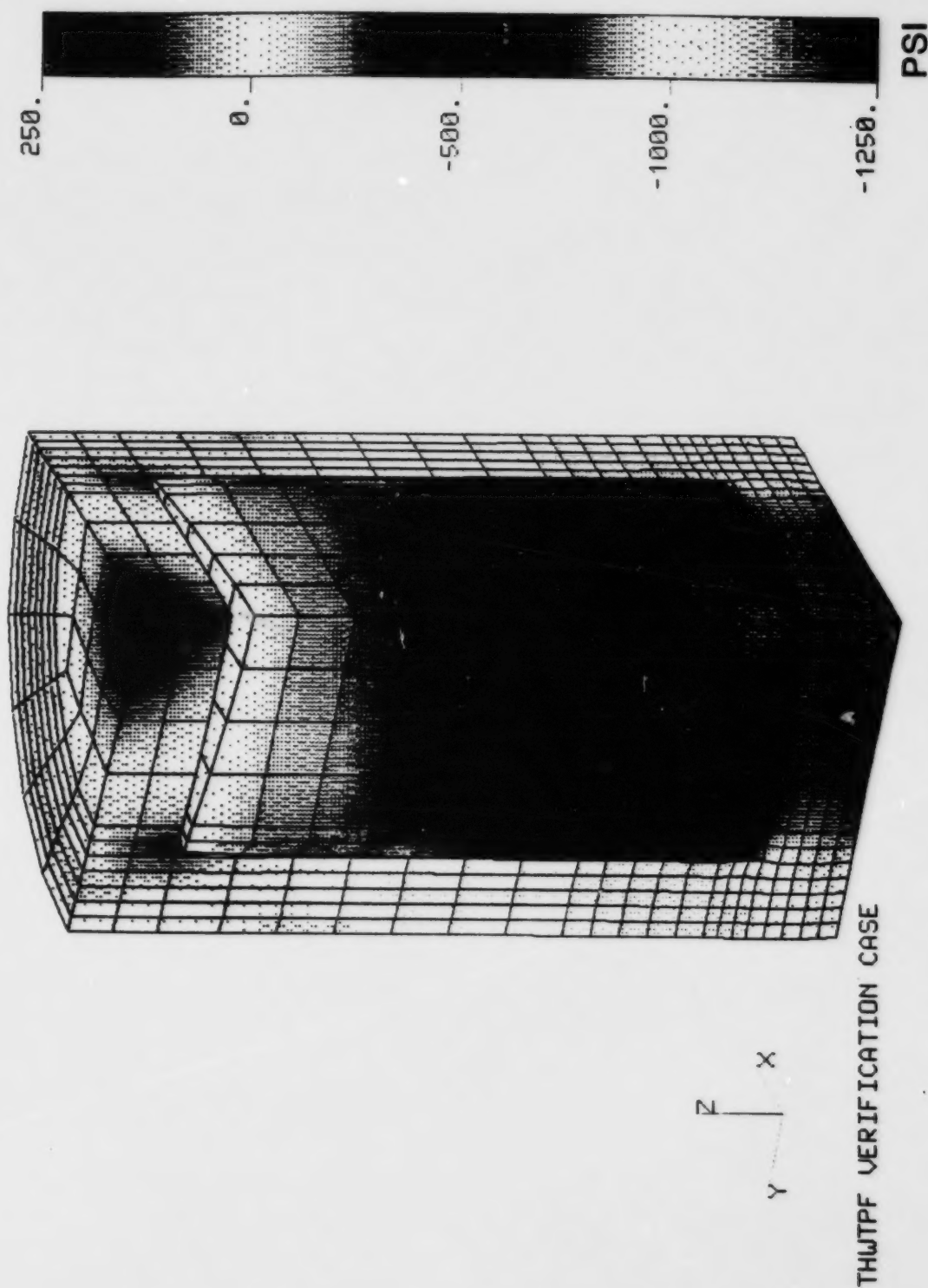
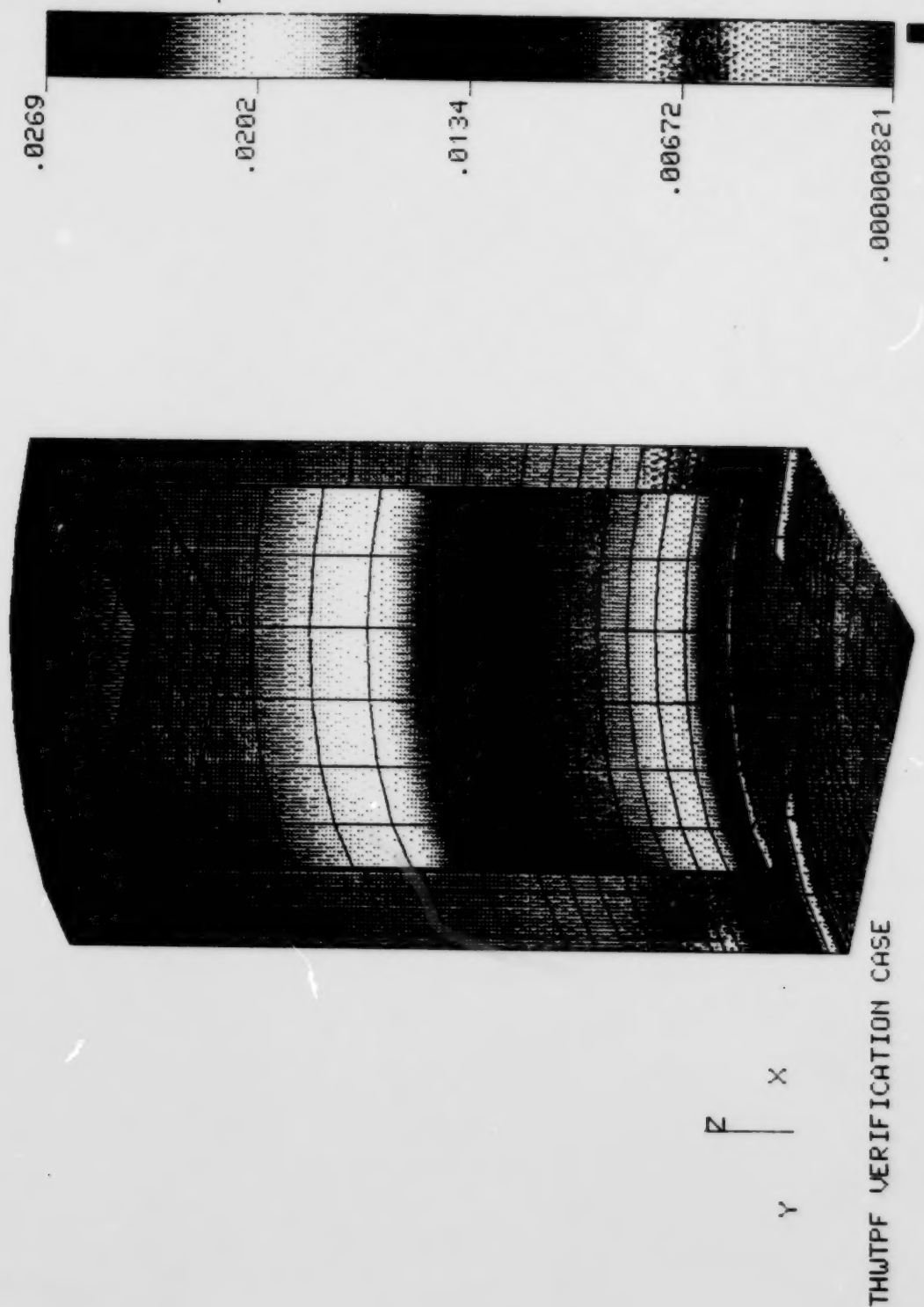


FIGURE 2.62.12
CASE 67: HYDRODYNAMIC STRESS DISTRIBUTION -
TIME = 7 ms





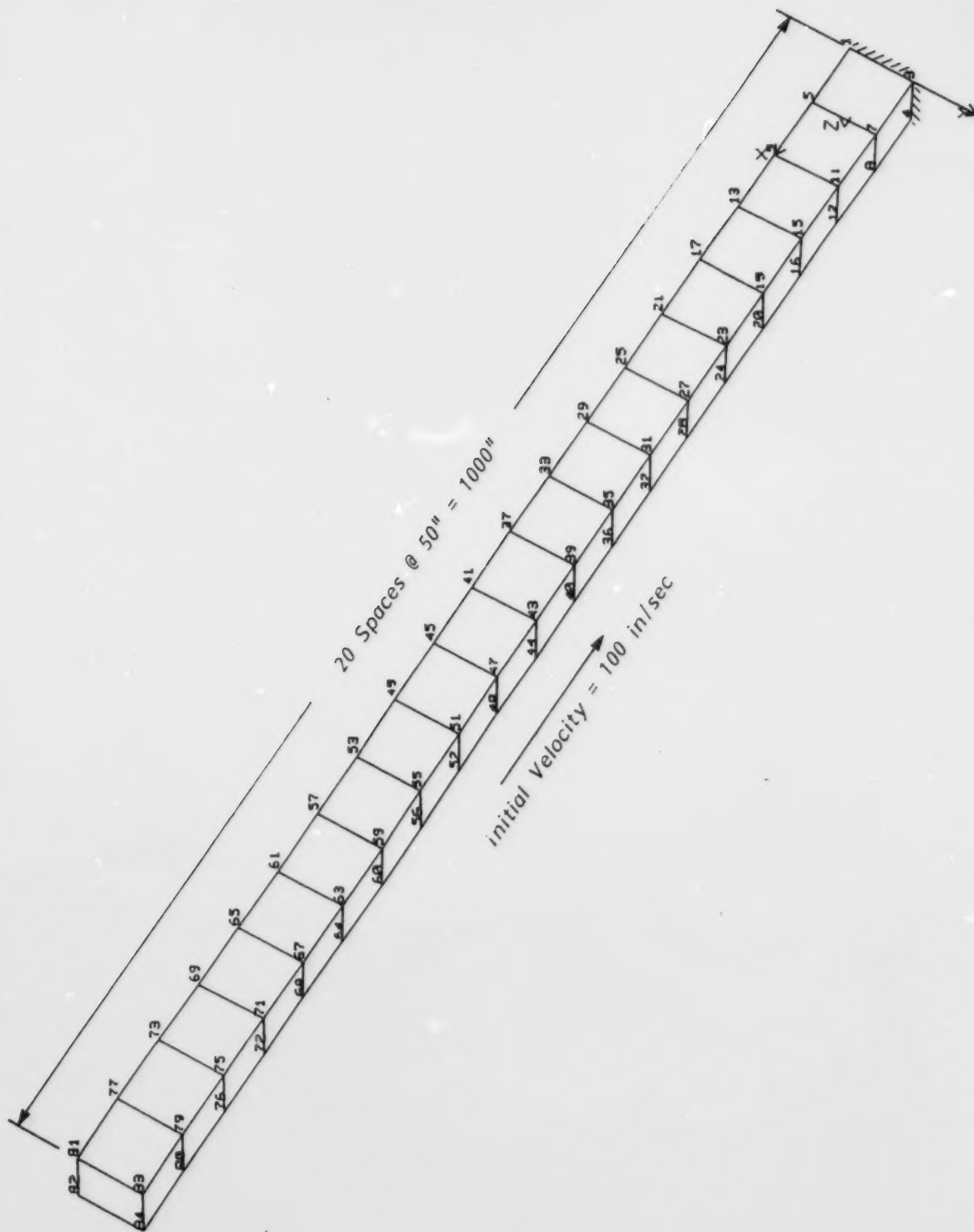


FIGURE 2.63.1
CASE 63 - FLUID HYDRODYNAMIC MATERIAL TESTING
H3DMAP MODEL

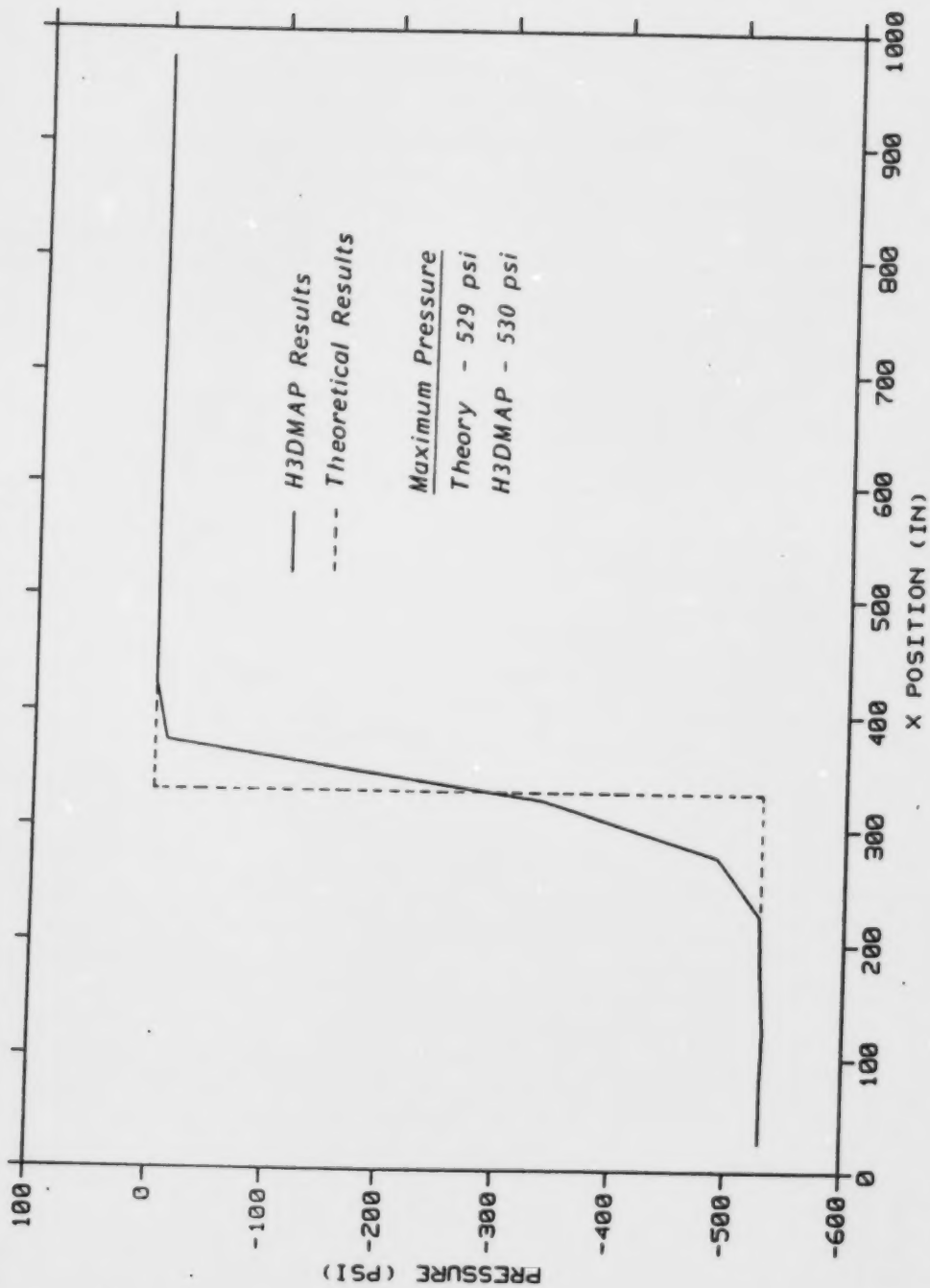


FIGURE 2.63.2

CASE 63: PRESSURE DISTRIBUTION IN THE PIPE AT TIME = 0.0058 SECOND

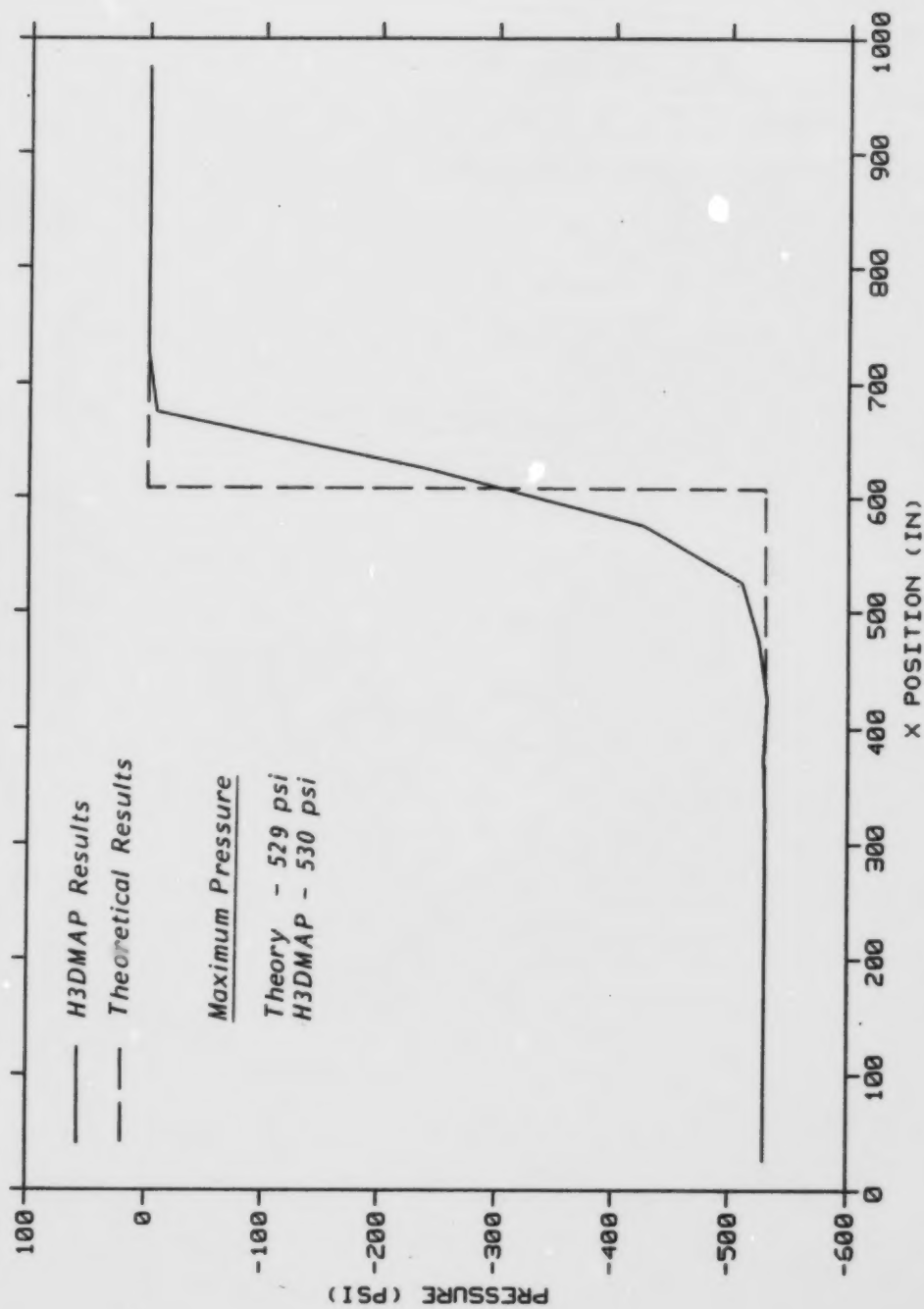


FIGURE 2.63.3

CASE 63: PRESSURE DISTRIBUTION IN THE PIPE AT TIME = 0.01077 SECOND

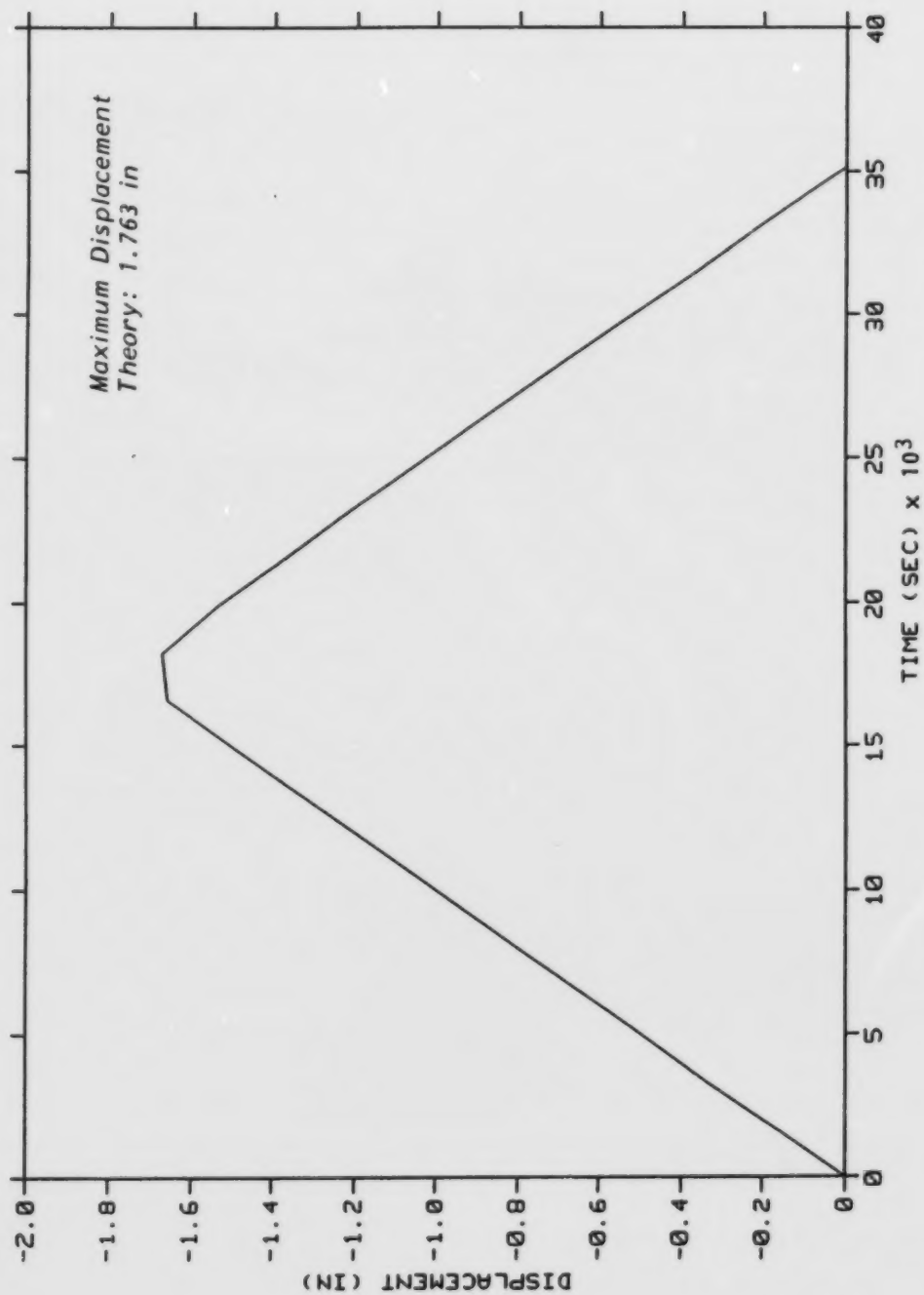


FIGURE 2.63.4

CASE 63 - DISPLACEMENT OF FREE SURFACE - NODE 84

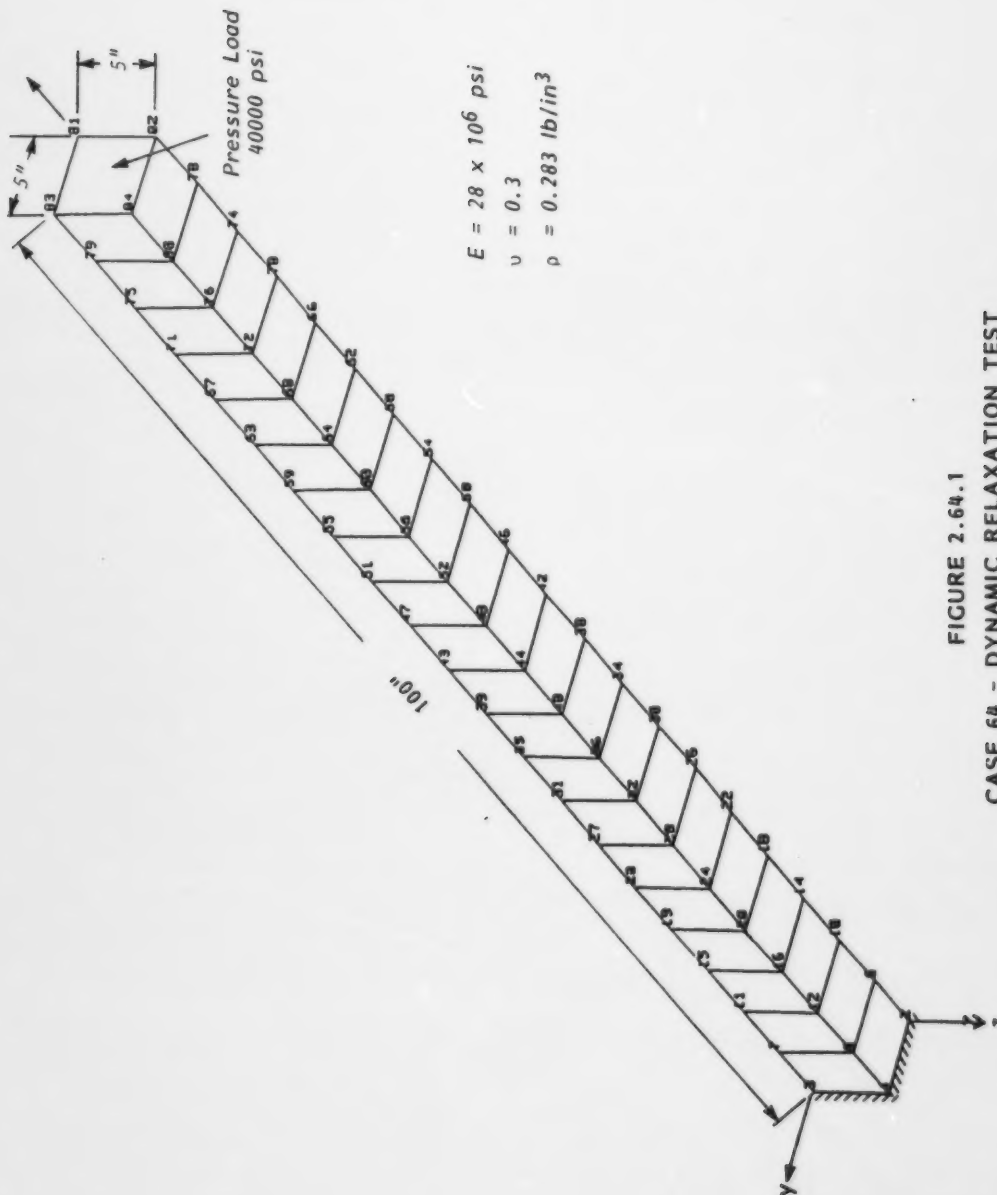


FIGURE 2.64.1
CASE 64 - DYNAMIC RELAXATION TEST
H3DMAP MODEL

181701-RD

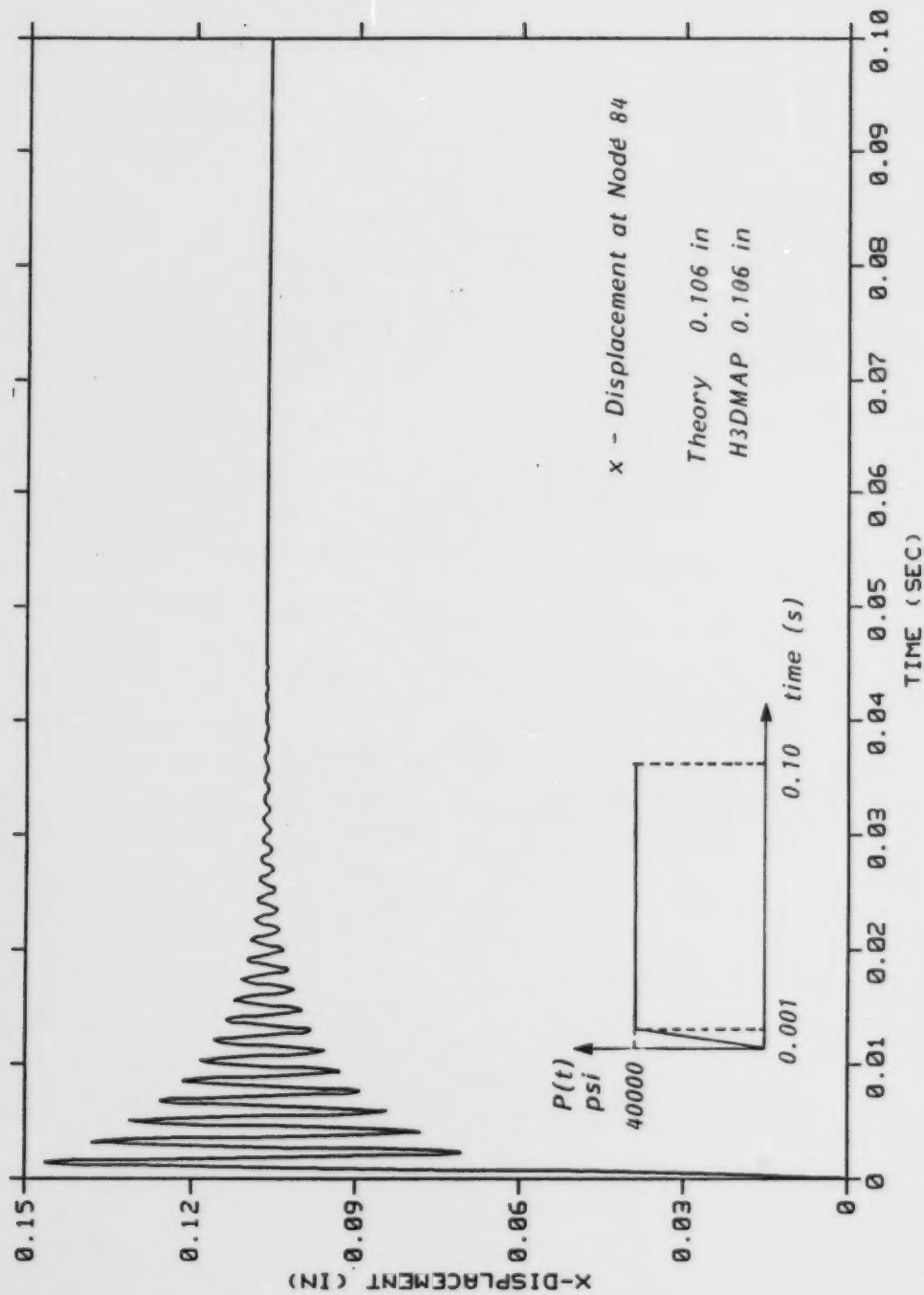


FIGURE 2.64.2

CASE 64: DISPLACEMENT RESPONSE OF FREE END (NODE 84)

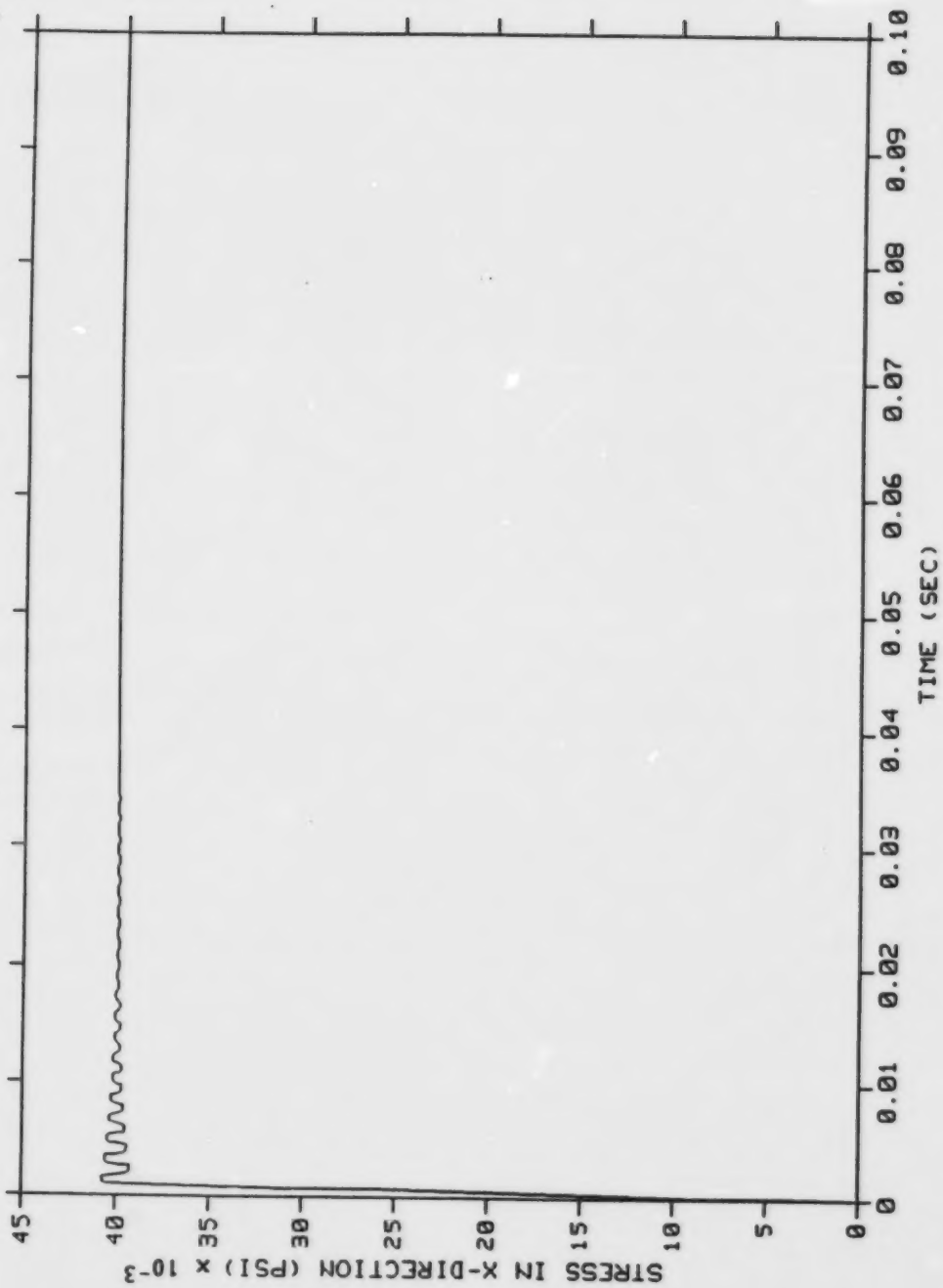


FIGURE 2.64.3
CASE 64: STRESS RESPONSE OF FREE END (ELEMENT 20)

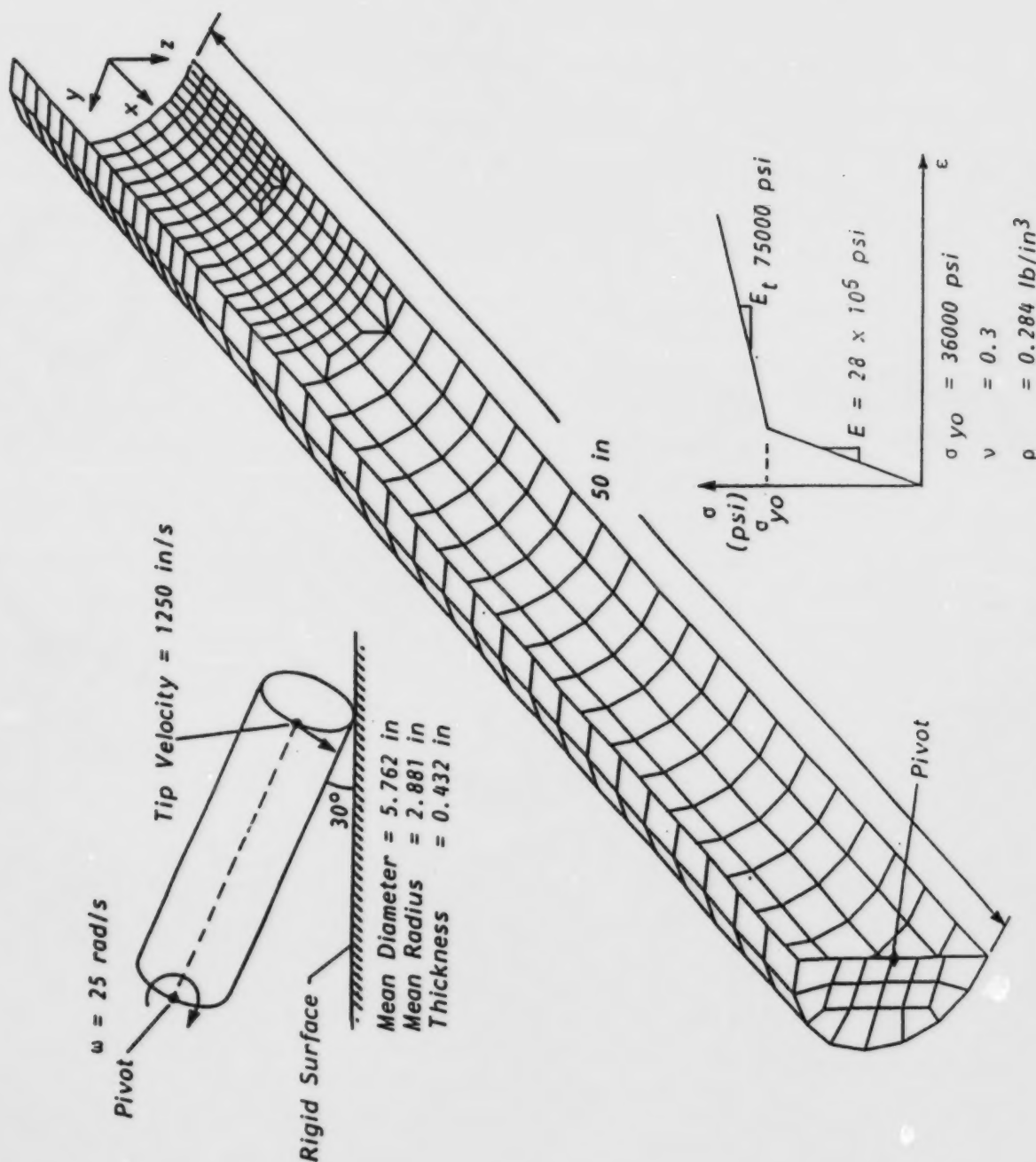


FIGURE 2.65.1

CASE 65: PIPE WHIP TEST - MODEL AND MATERIAL PROPERTIES

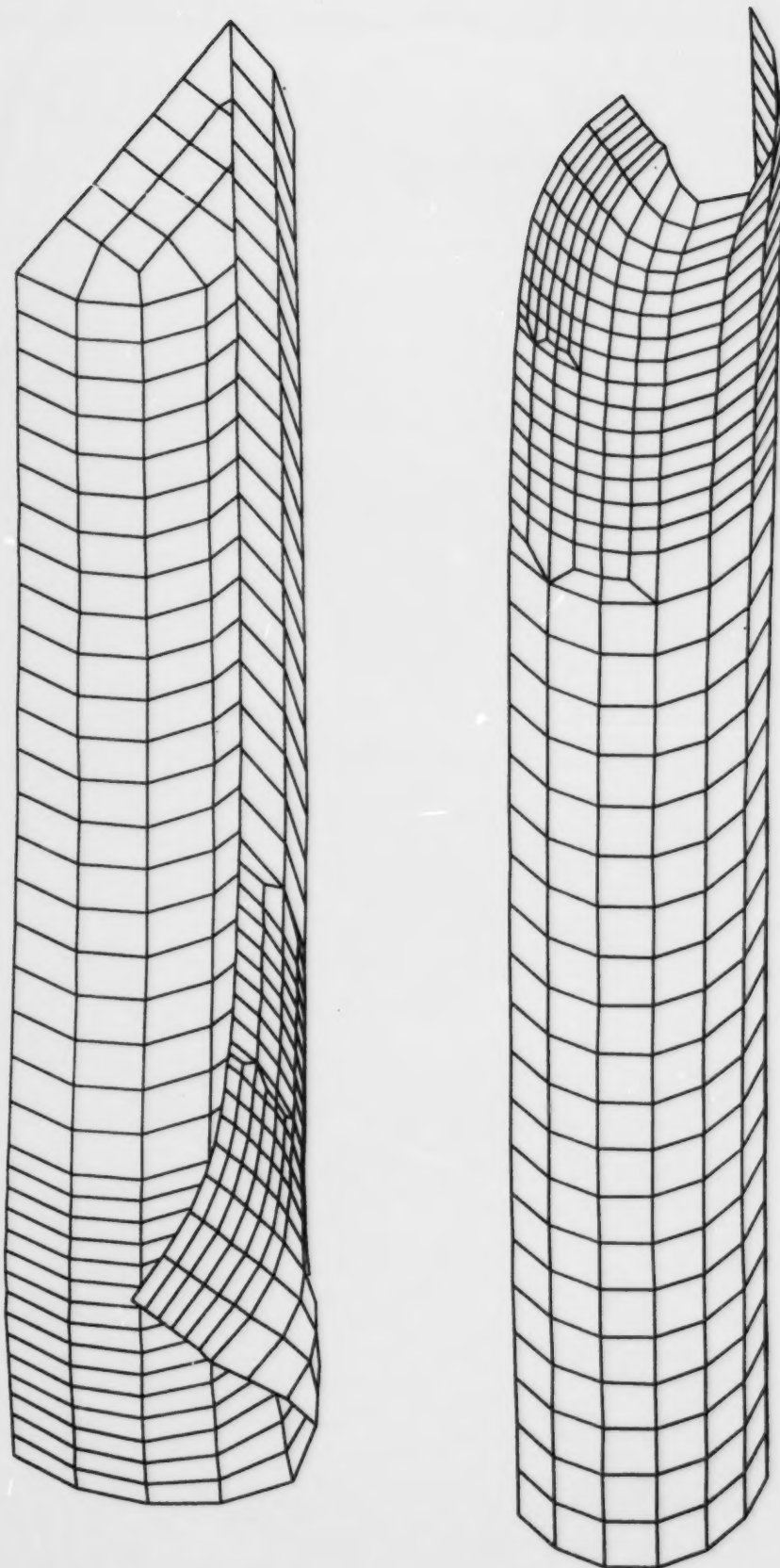


FIGURE 2.65.2
CASE 65: DEFORMED STRUCTURE - RESPONSE AT TIME 0.0035 SECOND

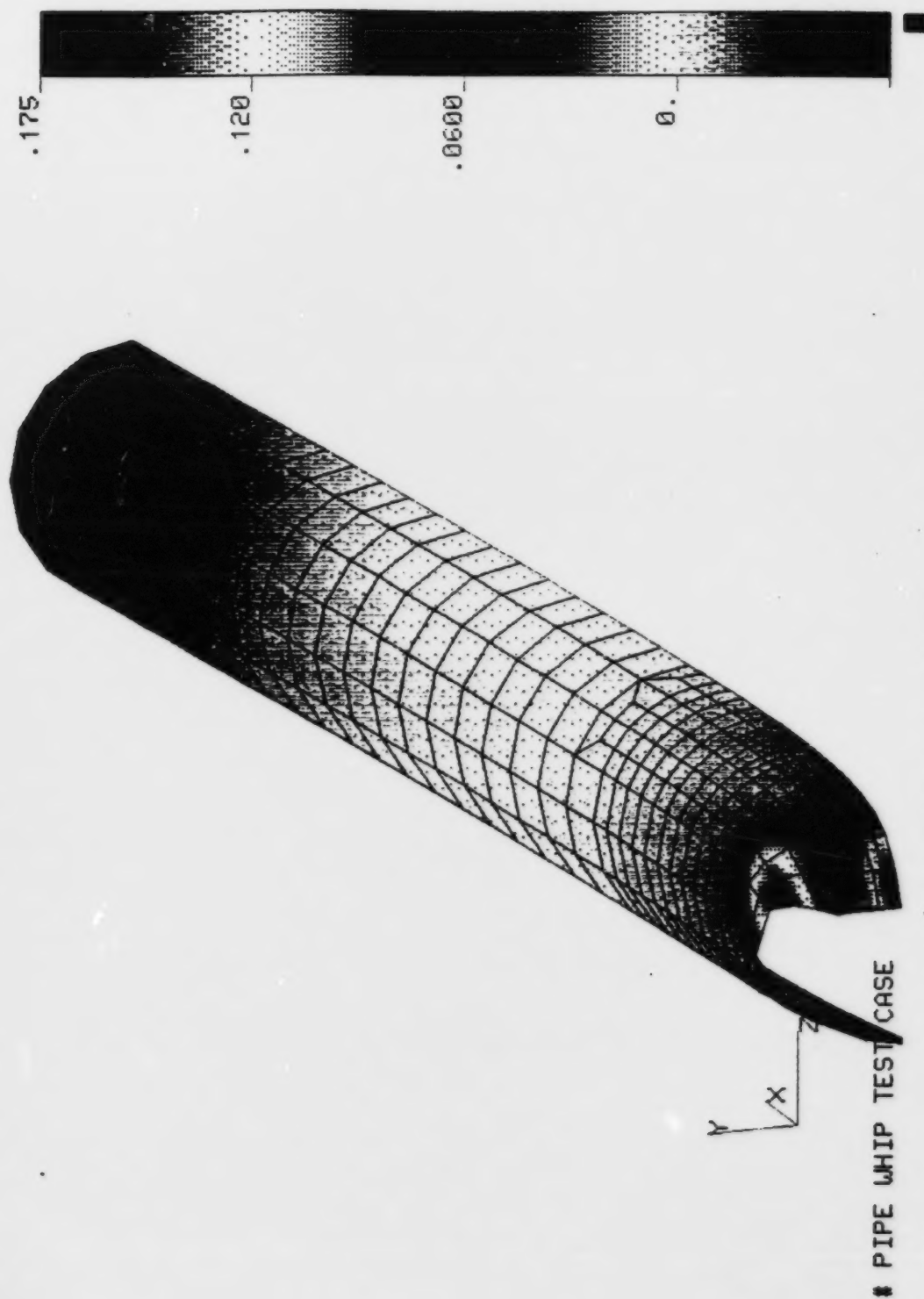


FIGURE 2.65.3
CASE 65: EFFECTIVE PLASTIC STRAIN - OUTER SHELL
TIME: 3.5 ms

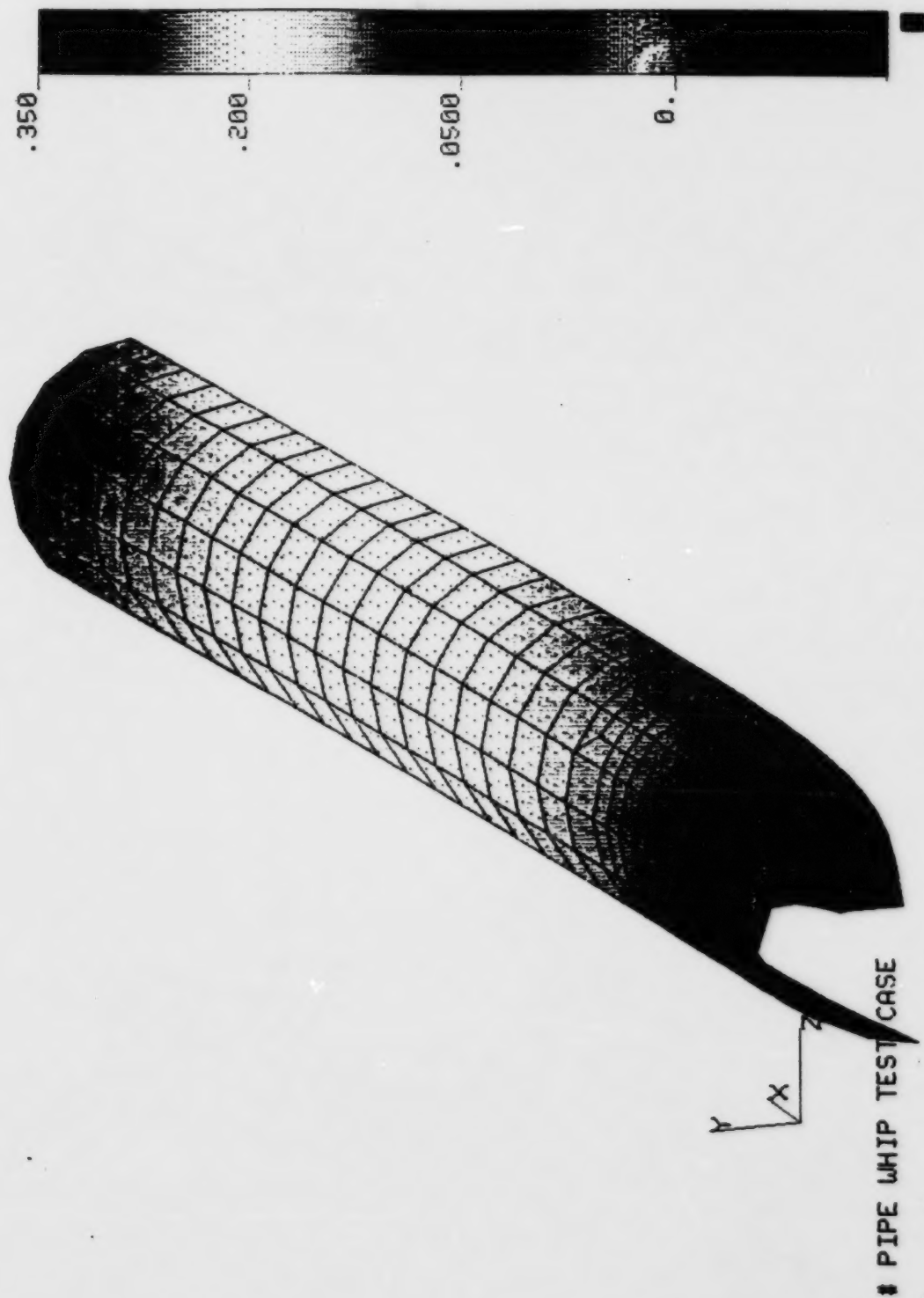


FIGURE 2.65.4
CASE 65: EFFECTIVE PLASTIC STRAIN - INNER SHELL
TIME: 3.5 ms

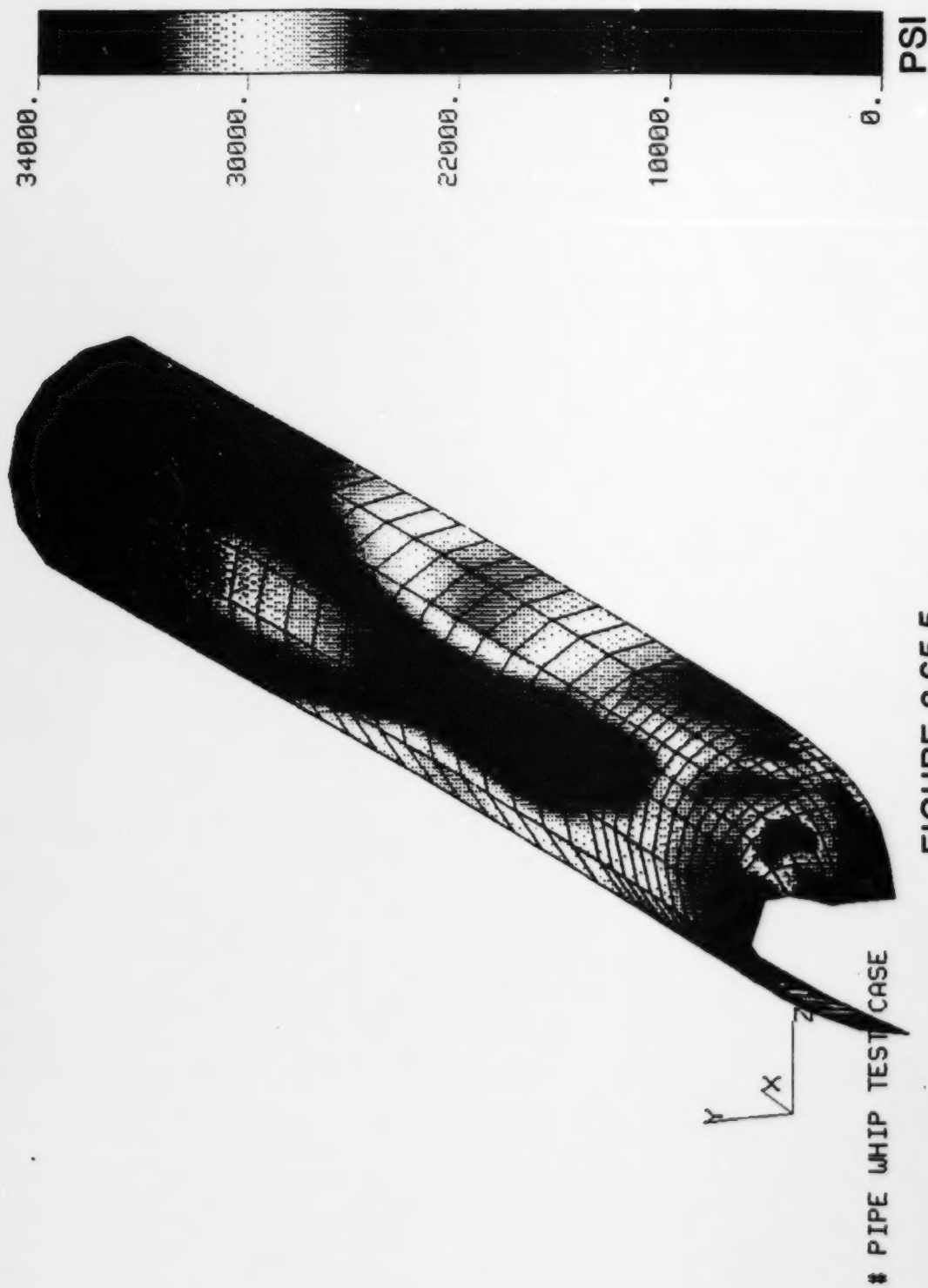


FIGURE 2.65.5
CASE 65: VON MISES STRESS - OUTER SHELL
TIME: 3.5 ms

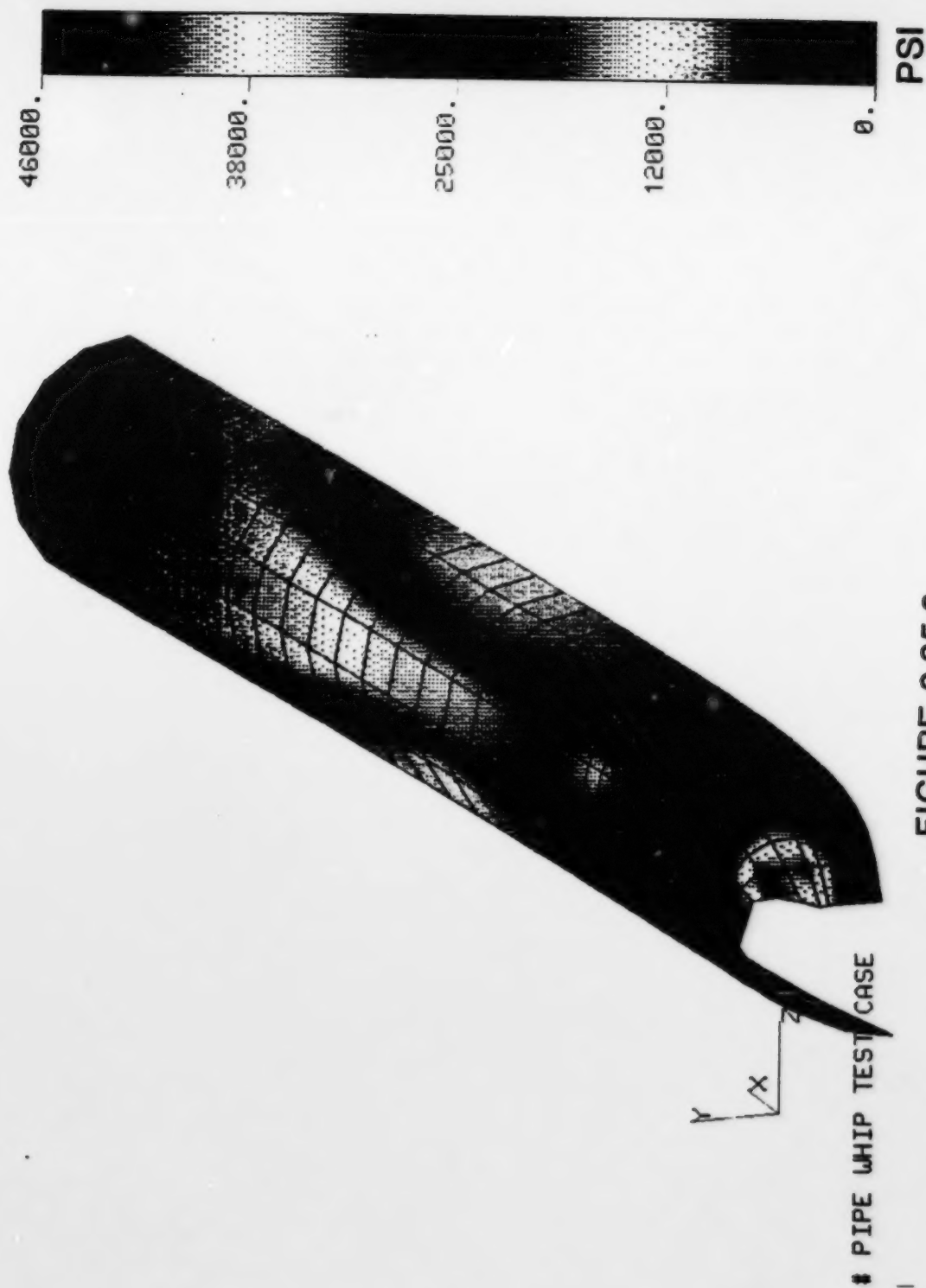


FIGURE 2.65.6
CASE 65: VON MISES STRESS - INNER SHELL
TIME: 3.5 ms



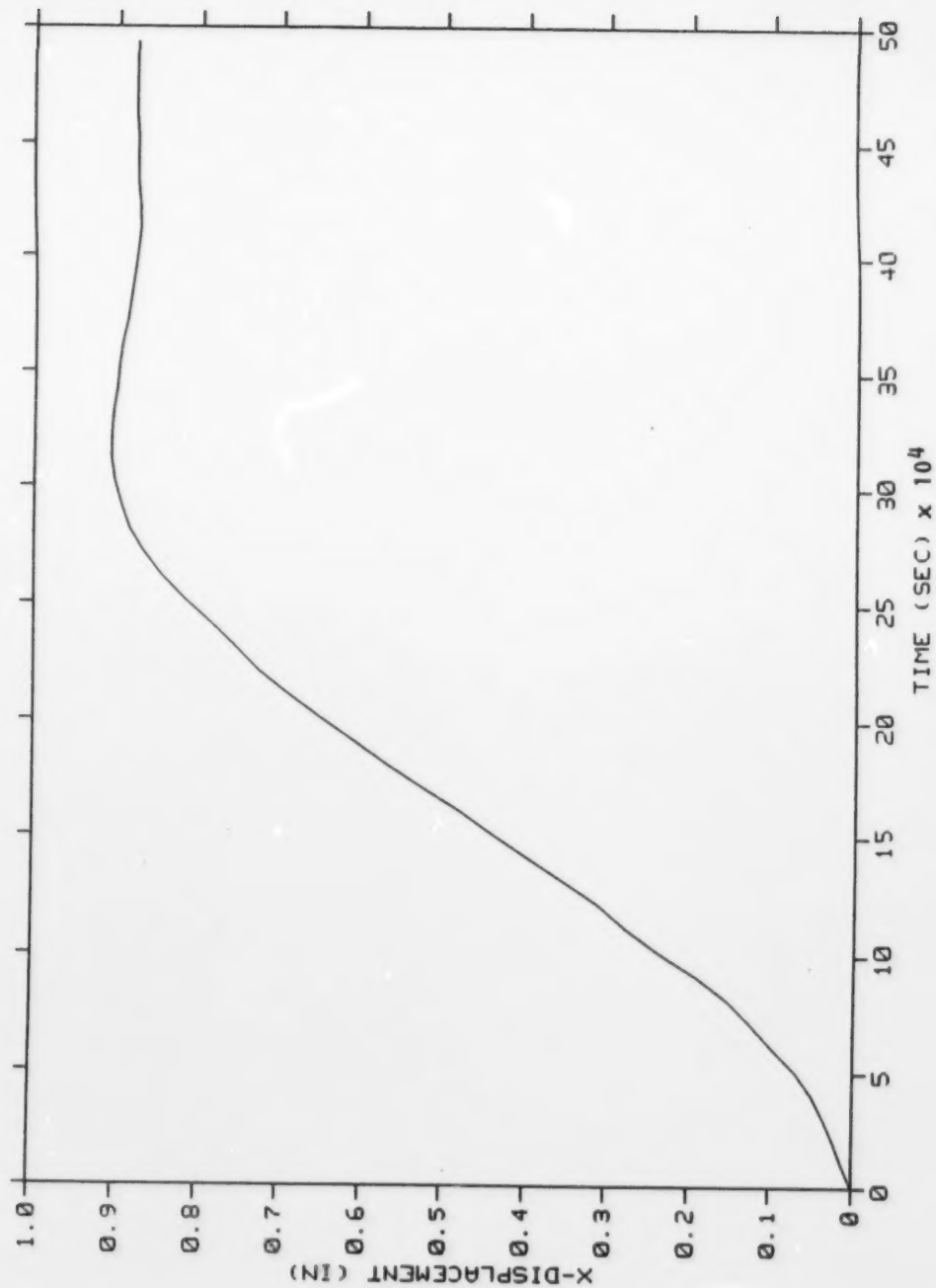


FIGURE 2.65.7
CASE 65: X-DISPLACEMENT TIME HISTORY OF NODE 1

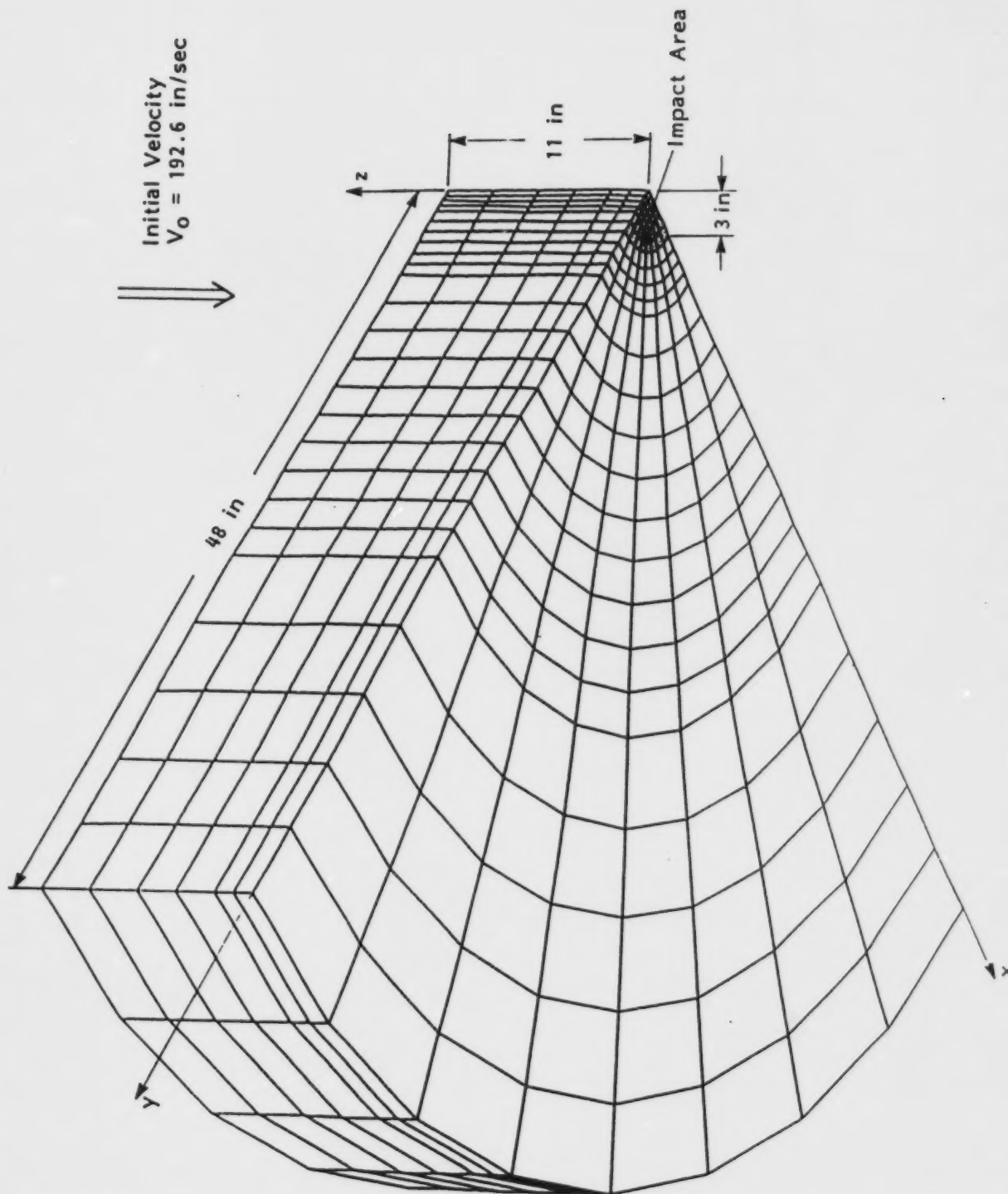


FIGURE 2.66.1
 CASE 66: TRITIATED D_2O TRANSPORT CONTAINER PUNCH TEST

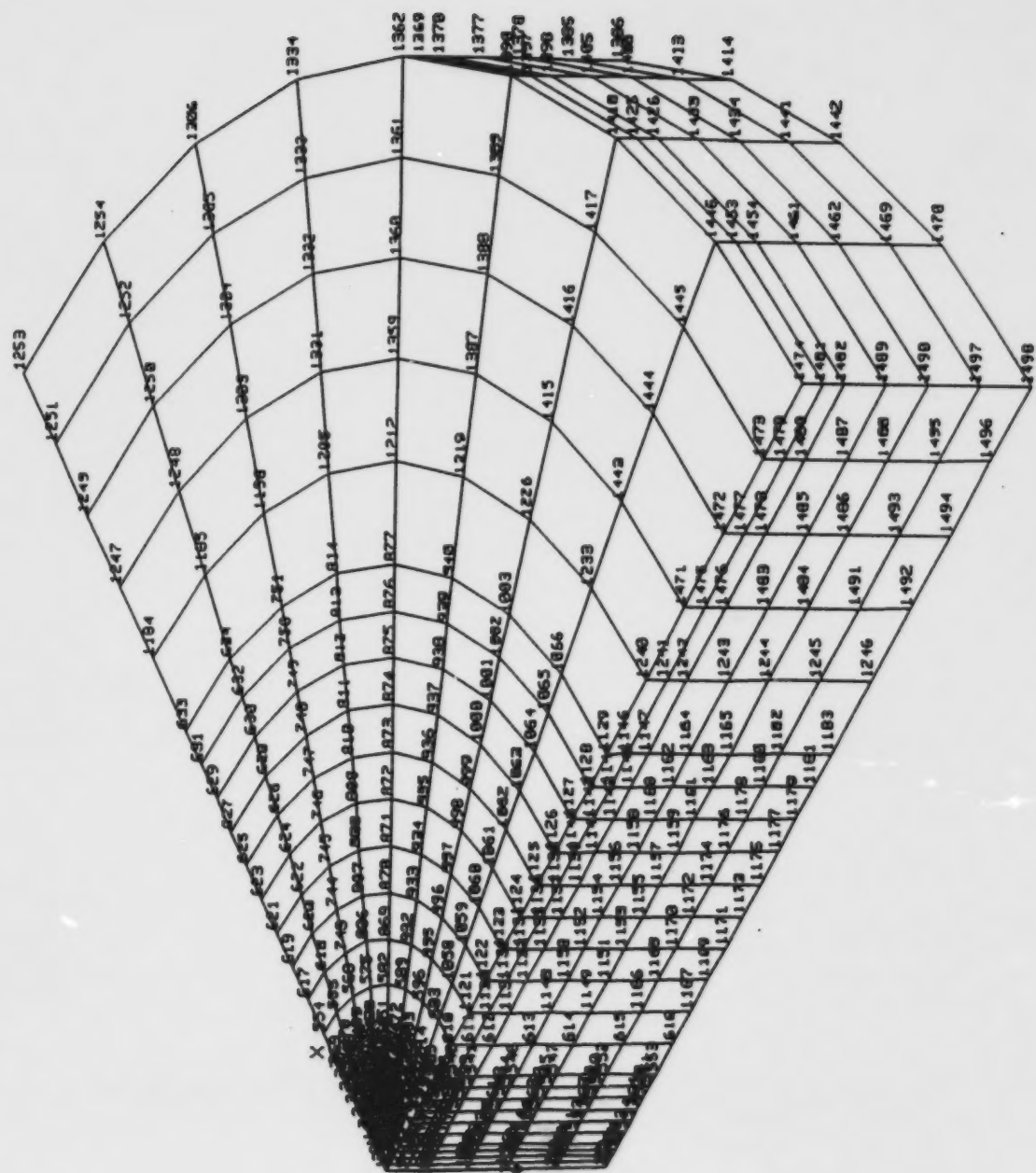


FIGURE 2.66.2
CASE 66: NODE NUMBERING

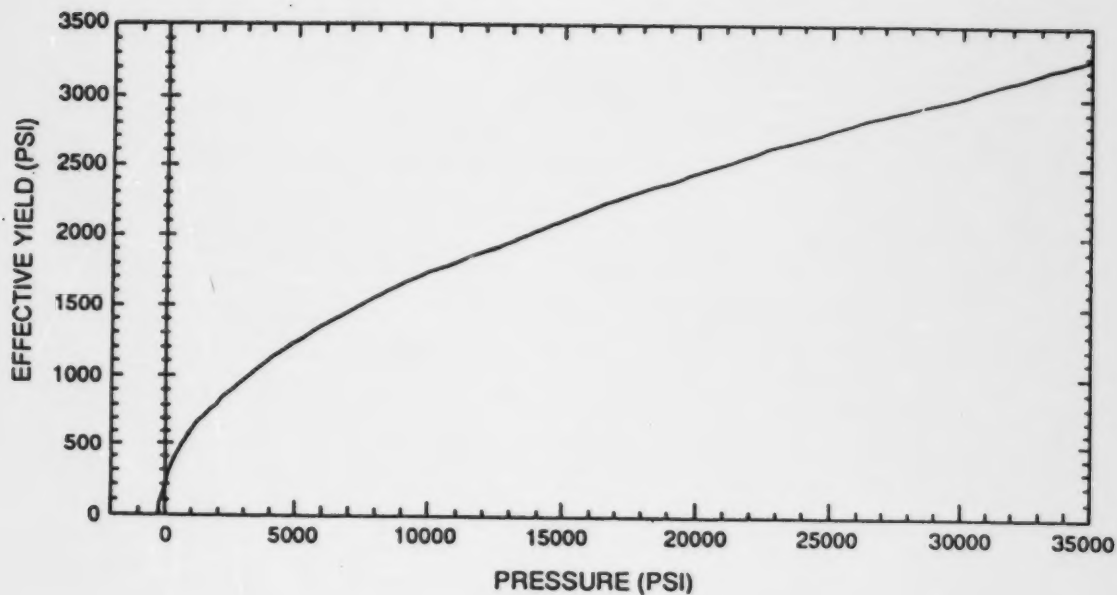


FIGURE 2.66.4
CASE 66: YIELD VS PRESSURE CURVE FOR
POLYURETHANE FOAM MODEL

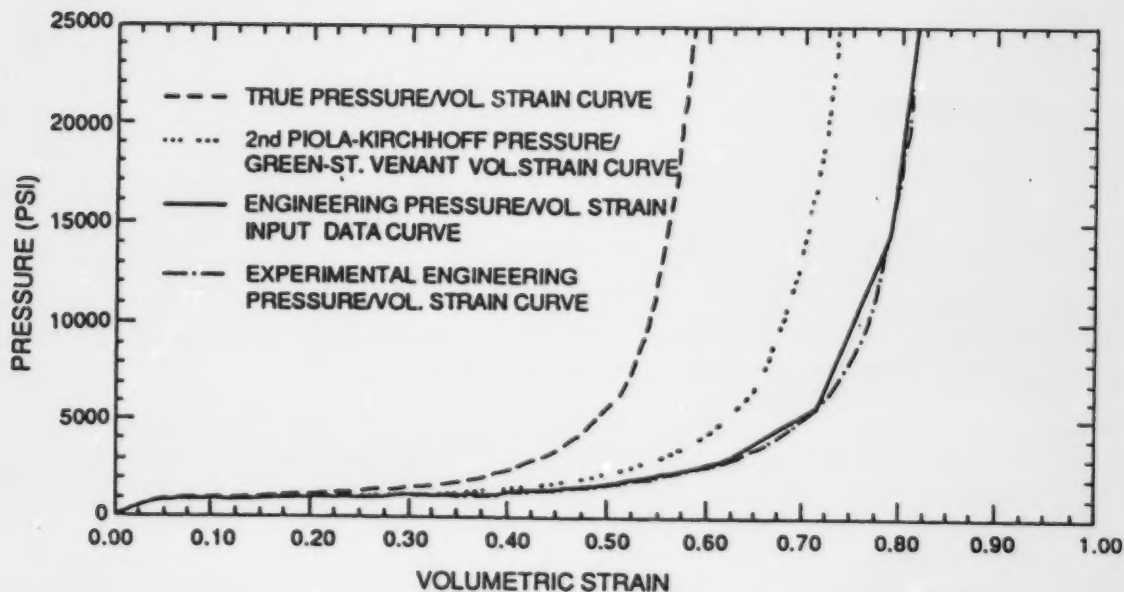


FIGURE 2.66.5
CASE 66: PRESSURE/VOLUMETRIC STRAIN CURVE FOR
POLYURETHANE FOAM MODEL

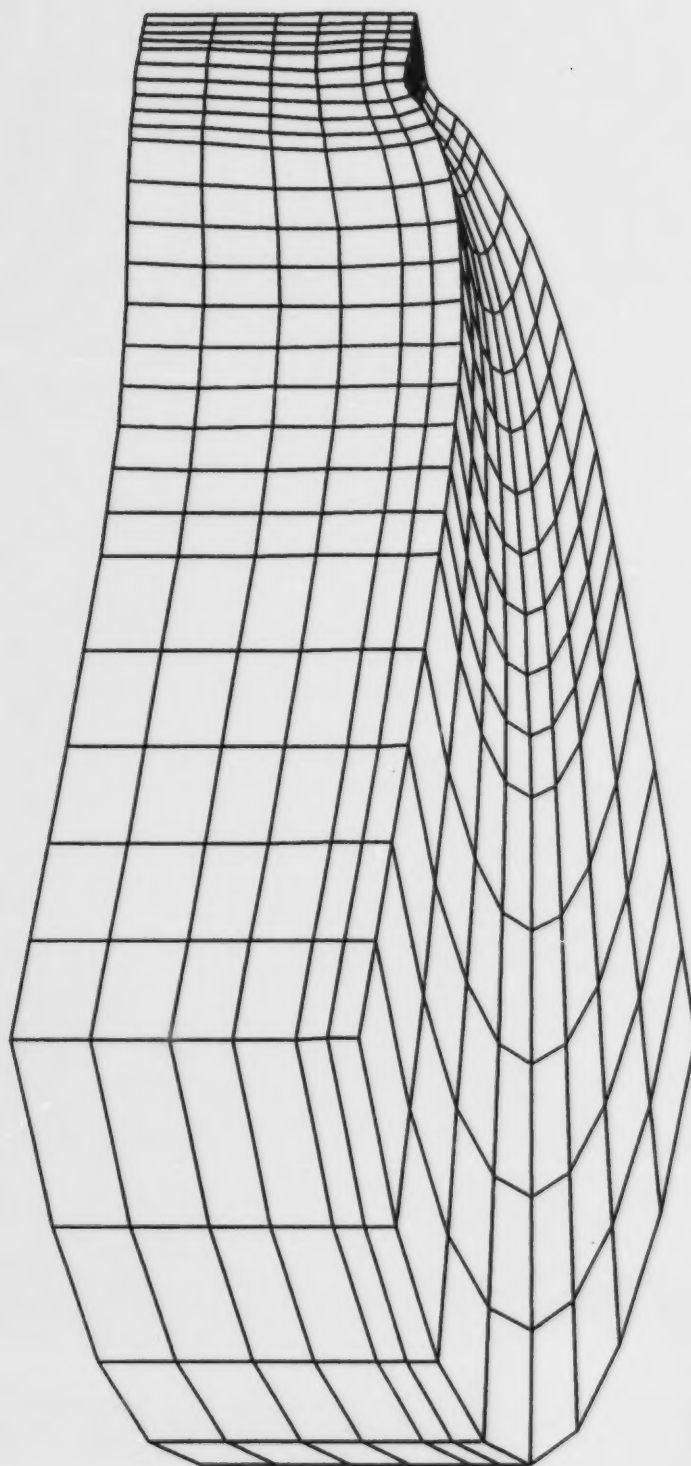


FIGURE 2.66.6
CASE 66 - DEFORMED GEOMETRY - RESPONSE AT TIME = 0.028 SECOND

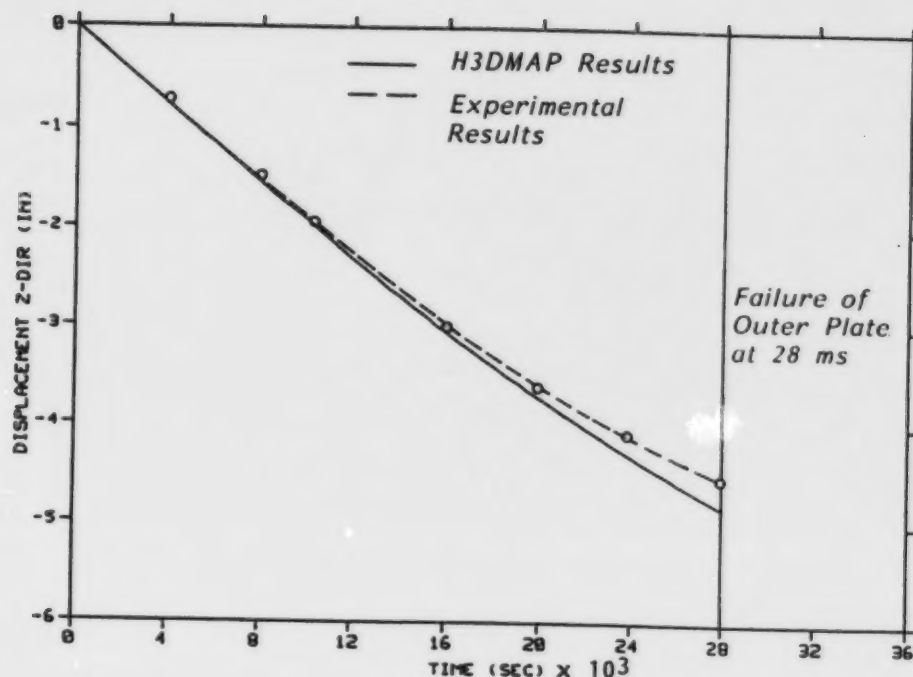


FIGURE 2.66.7

CASE 66: VERTICAL DISPLACEMENT AT THE PERIPHERY (NODE 1474)

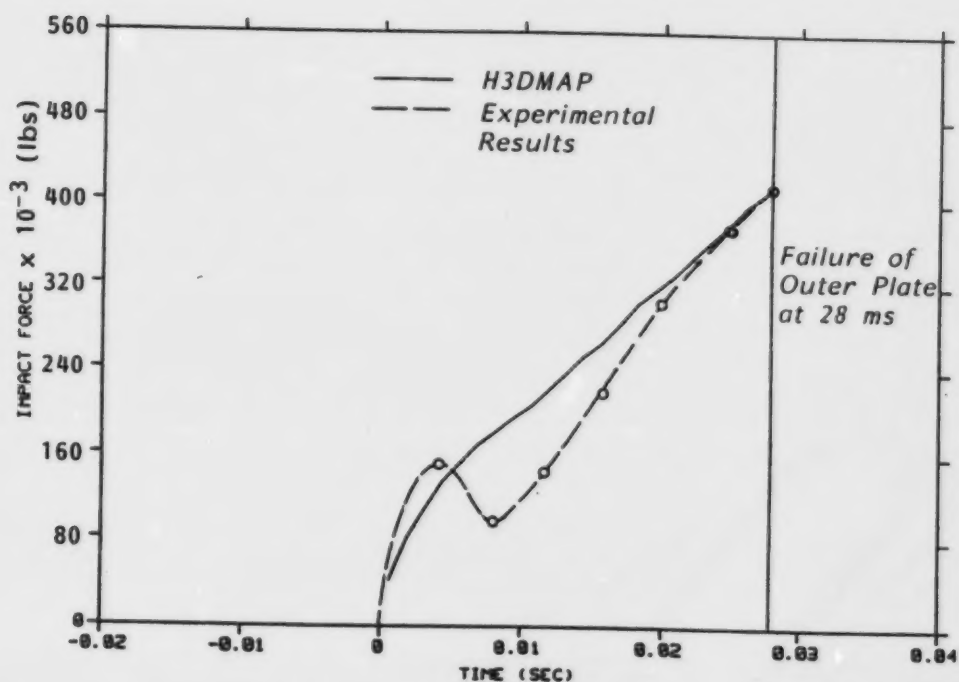


FIGURE 2.66.8

CASE 66: IMPACT FORCE TIME HISTORY

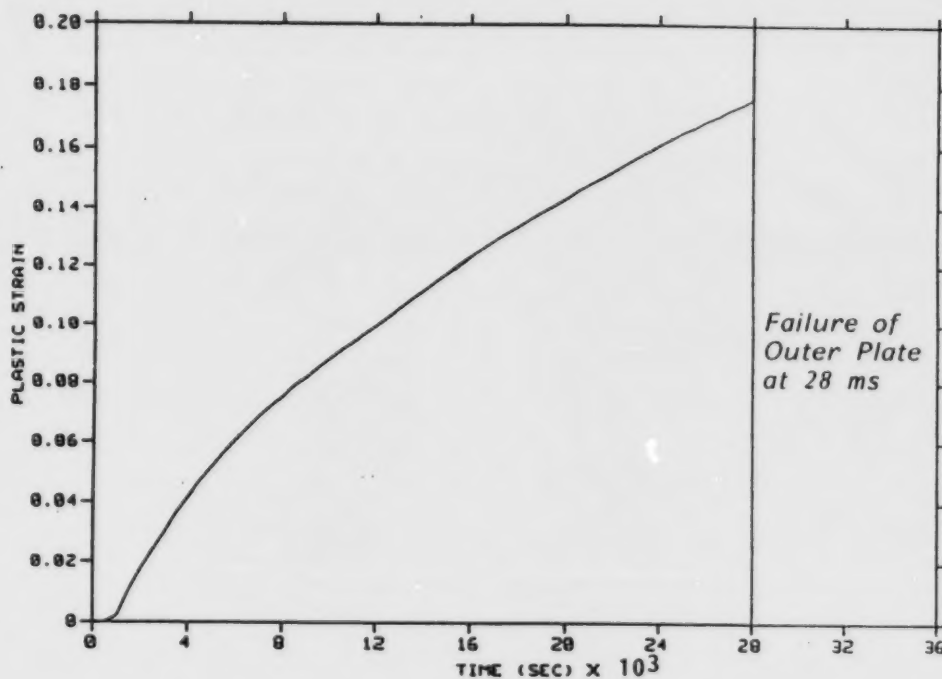


FIGURE 2.66.9

CASE 66: PLASTIC STRAIN IN SHELL ELEMENT
NO 10, SAMPLING PT No. 3
(See Figure 2.66.3)

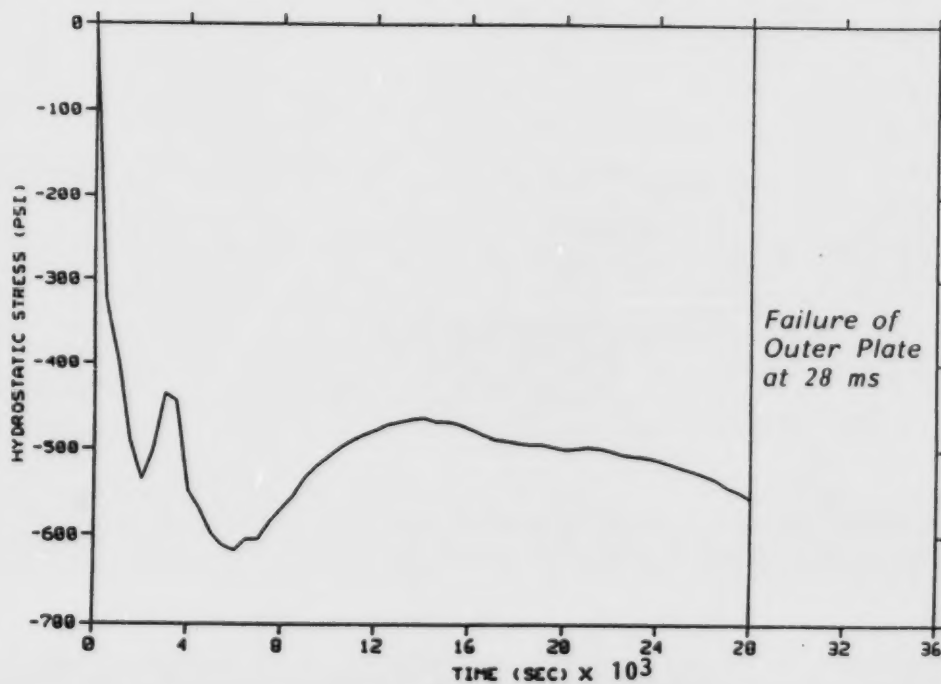


FIGURE 2.66.10

CASE 66: HYDROSTATIC STRESS IN SOLID ELEMENT NO 417
(See Figure 2.66.3)

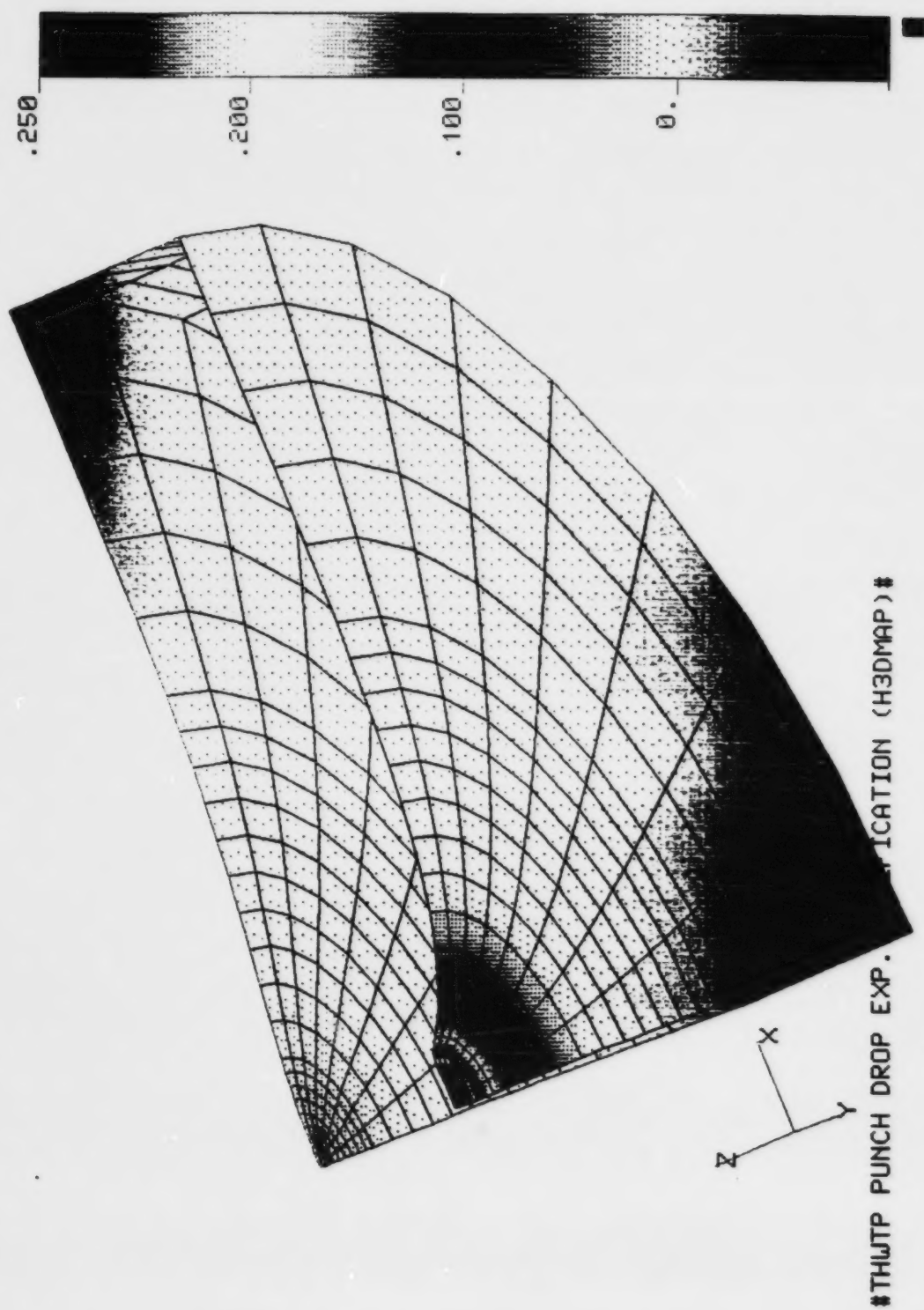
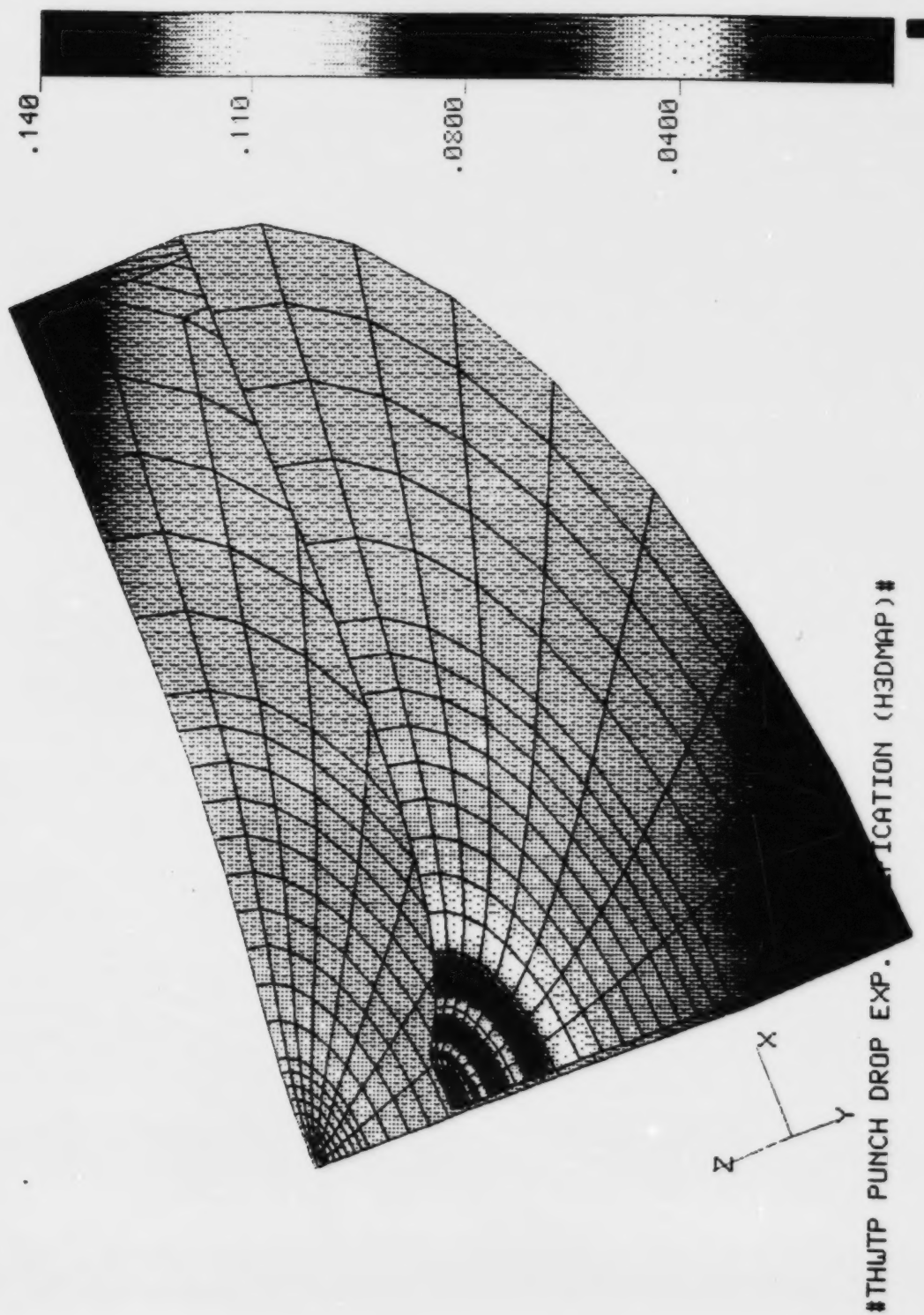


FIGURE 2.66.11
CASE 66: PLASTIC STRAIN IN METAL SHEET
OUTER SHELL - TIME: 28 ms



#THWTP PUNCH DROP EXP. ...ICATION (H3DMAP)#

FIGURE 2.66.12
CASE 66: PLASTIC STRAIN IN METAL SHEET
INNER SHELL - TIME: 28 ms

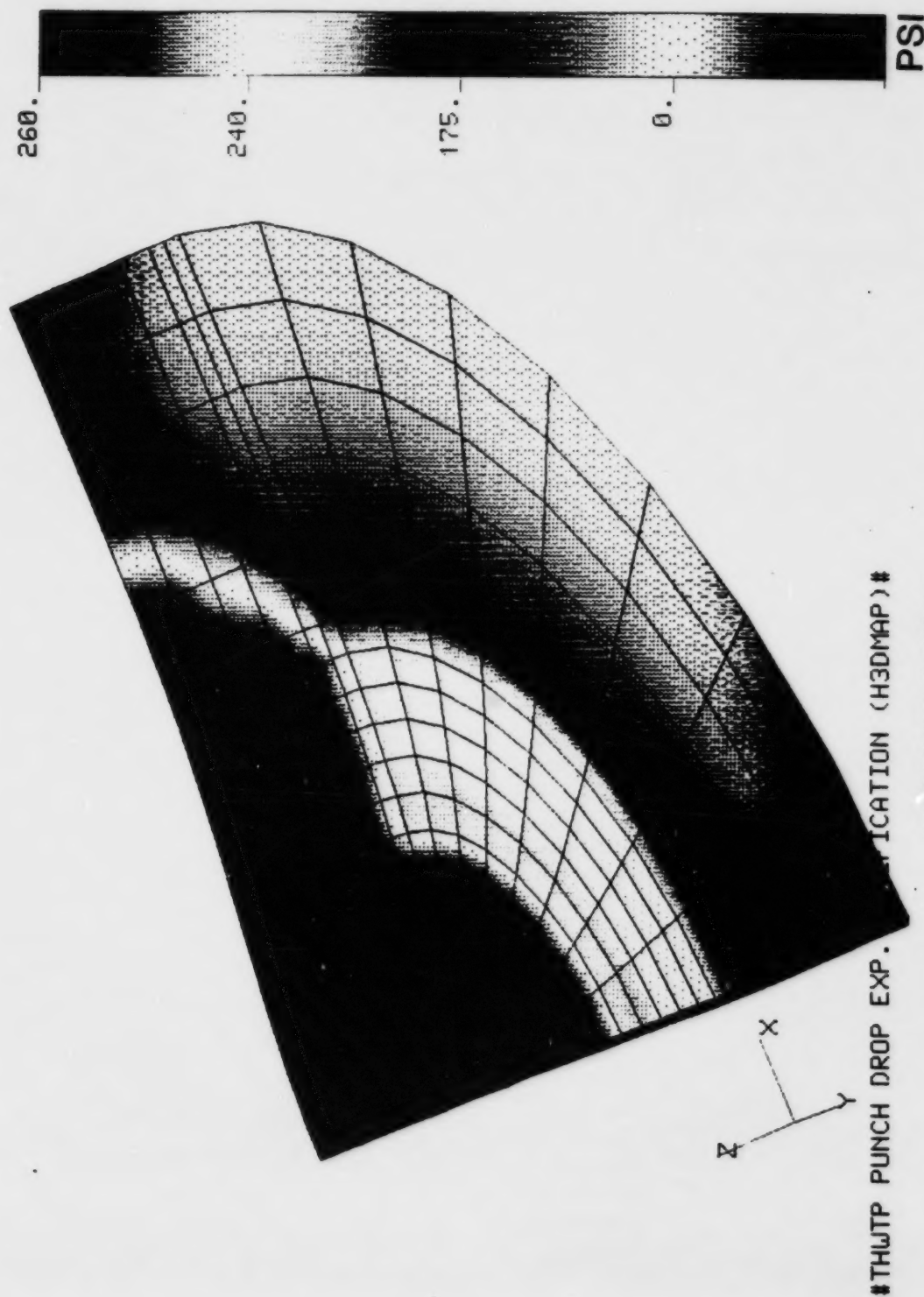


FIGURE 2.66.13
CASE 66: VON MISES STRESS IN FOAM
TIME: 28 ms

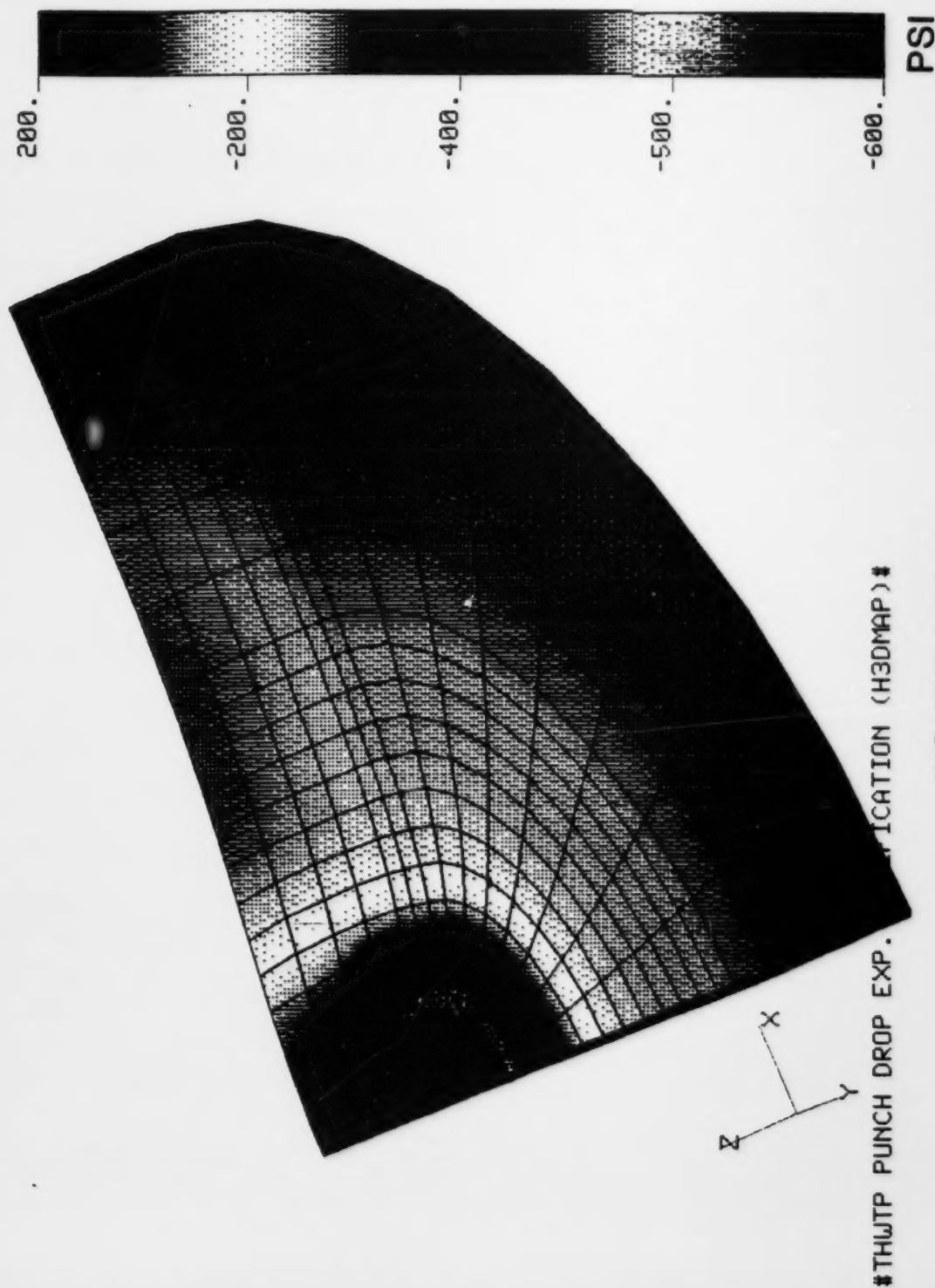


FIGURE 2.66.14
CASE 66: HYDROSTATIC STRESS IN FOAM
TIME: 28 ms

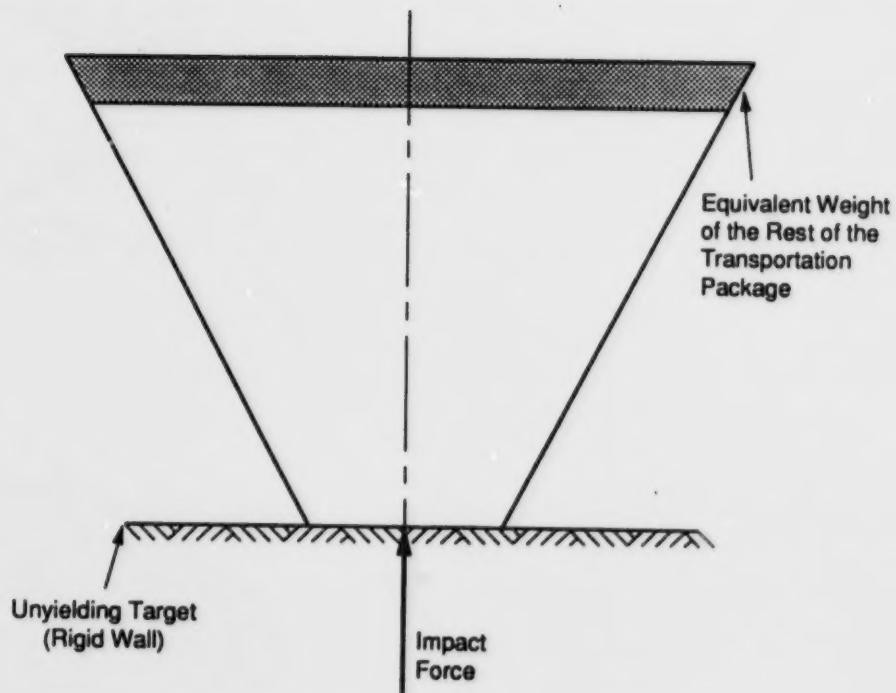


FIGURE 2.67.1

CASE 67: SIMPLIFIED ANALYTICAL MODEL

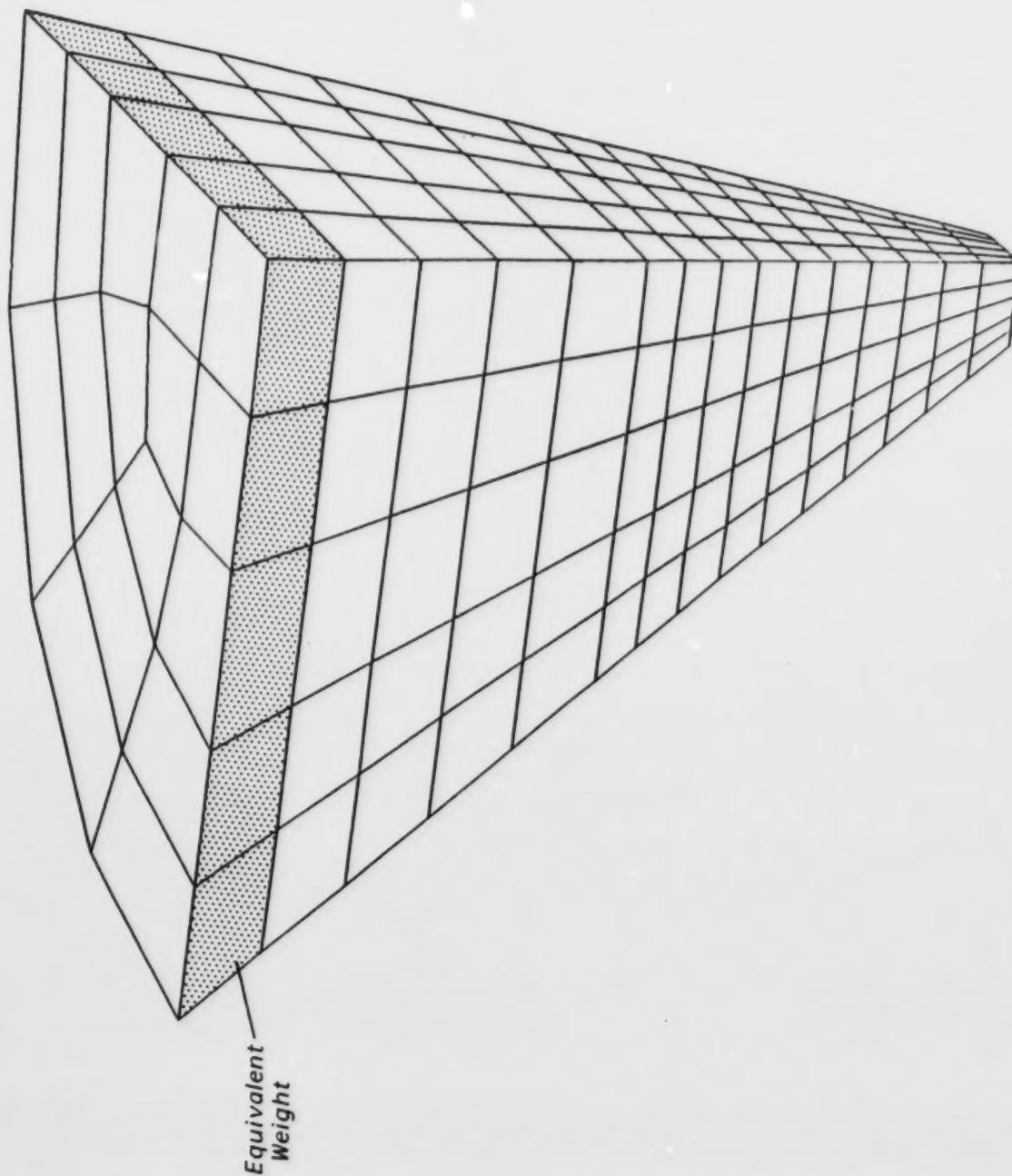


FIGURE 2.67.2
CASE 67: H3DMAP MODEL

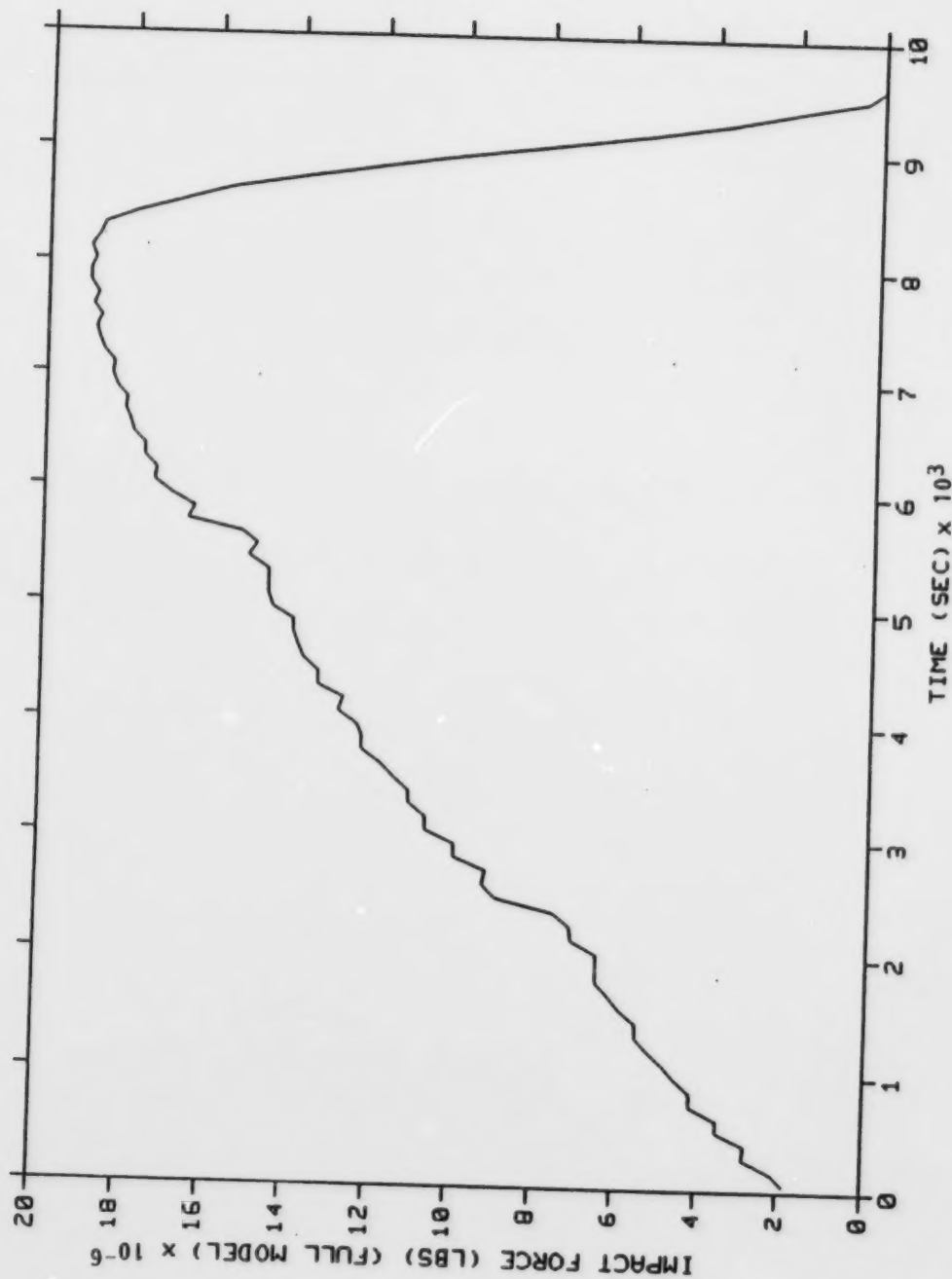


FIGURE 2.67.3
CASE 67: IMPACT FORCE VS TIME

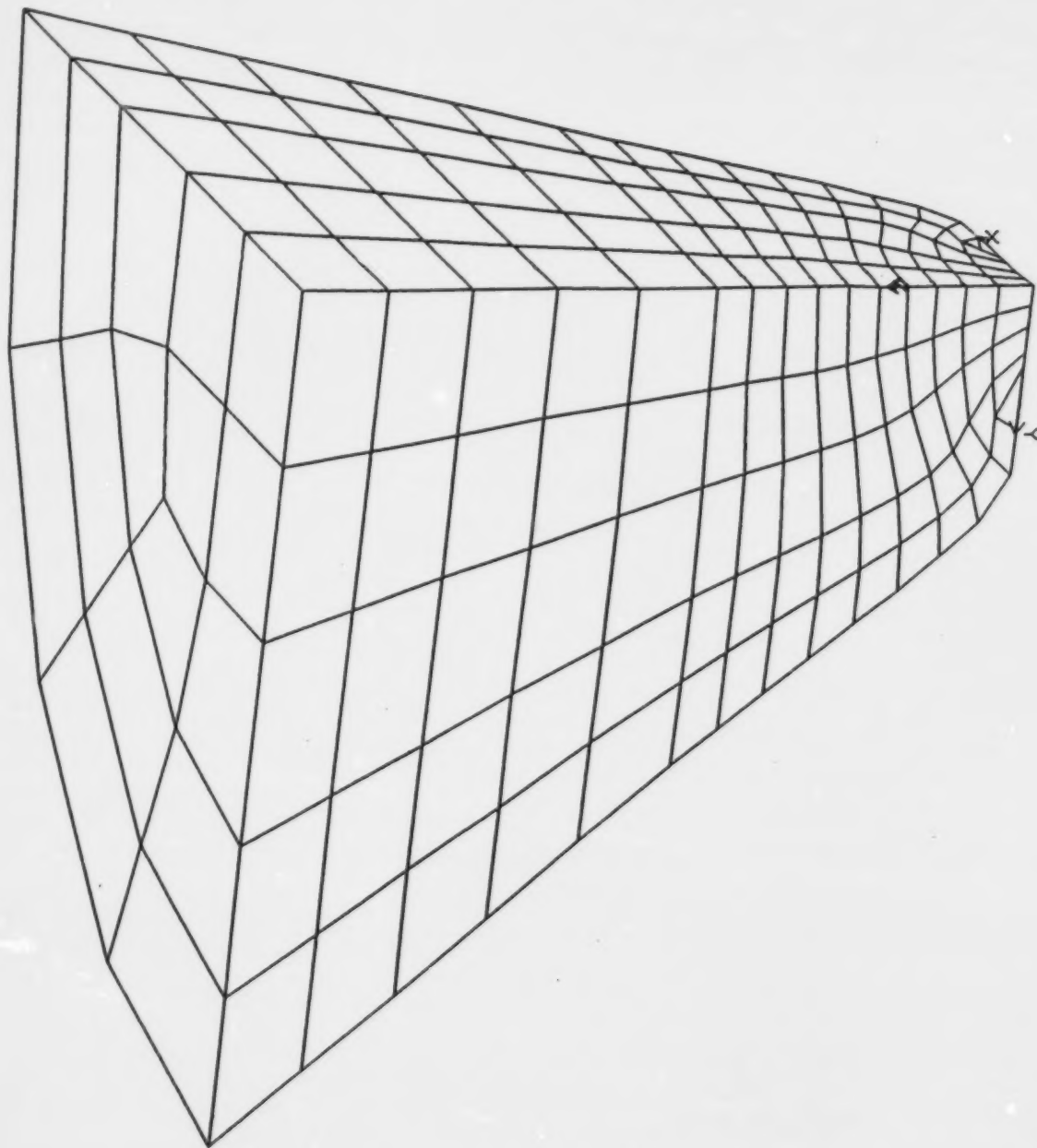


FIGURE 2.67.4
CASE 67: RESPONSE AT TIME = 8 ms

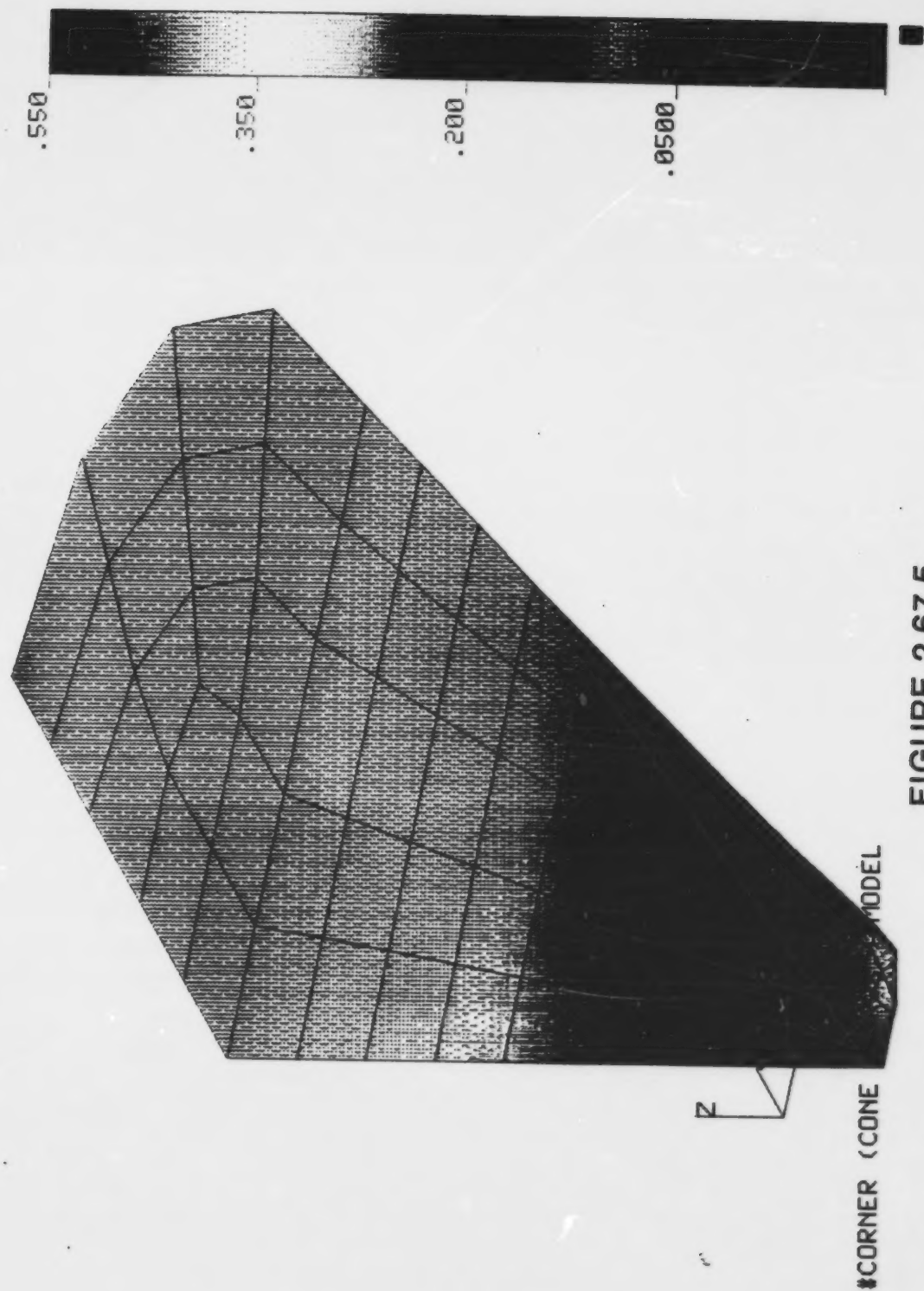
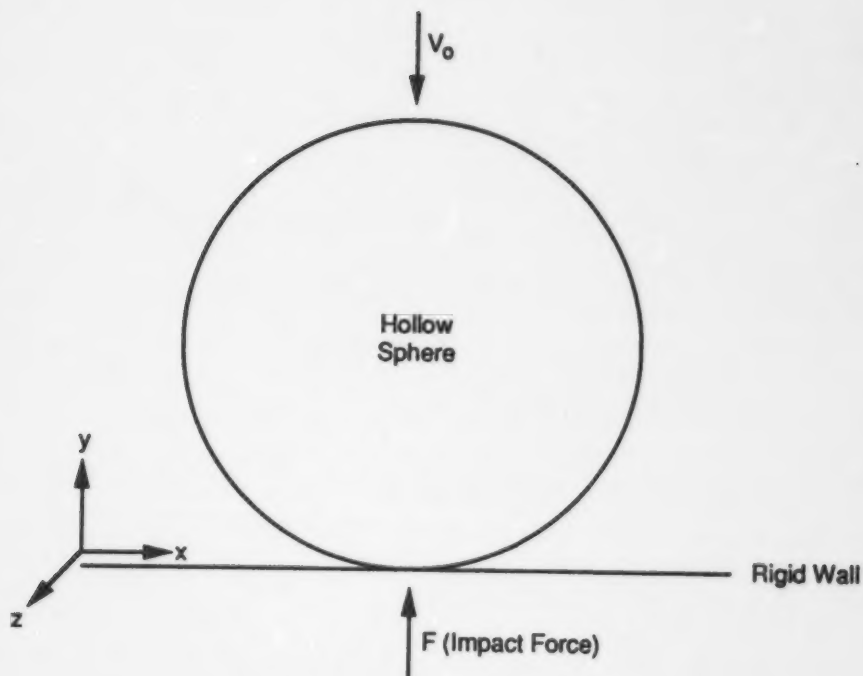


FIGURE 2.67.5
CASE 67: PLASTIC STRAIN DISTRIBUTION
TIME: 8 ms



Geometry

Outside Radius = 0.3048 m
Thickness = 0.01588 m

Material (Type 304SS)

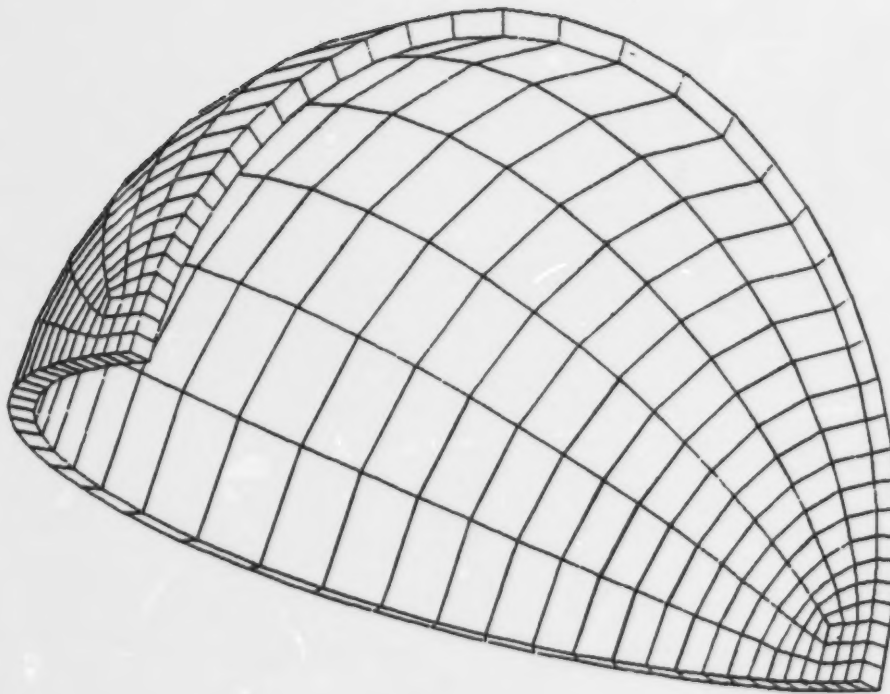
Elastic Modulus = 192 GPa
Hardening Modulus = 1.92 GPa
Yield Stress = 340 MPa
Poisson Ratio = 0.3

Initial Condition

Initial Velocity = 117.6 m/s

FIGURE 2.68.1

CASE 68: PROBLEM DESCRIPTION FOR THE SPHERE IMPACT ANALYSIS



Number of Nodes = 526
Number of Elements = 222

FIGURE 2.68.2

CASE 68: FINITE ELEMENT MODEL FOR THE SPHERE IMPACT ANALYSIS

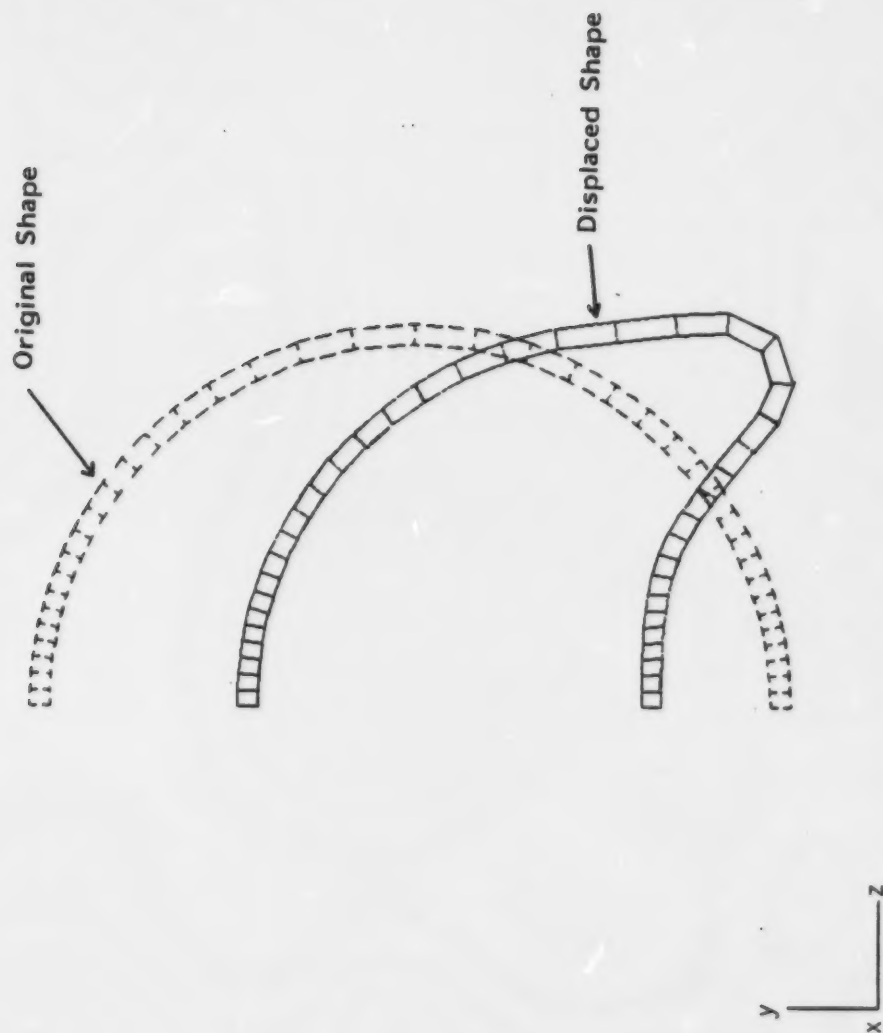
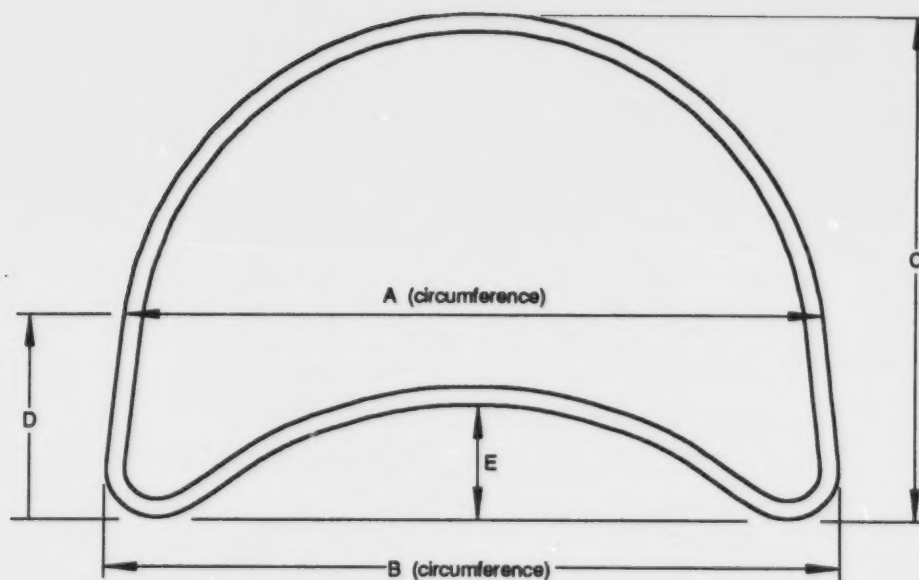


FIGURE 2.68.3

CASE 68: CROSS-SECTIONAL VIEW OF THE SPHERE DEFORMATION AT TIME $t = 3 \text{ ms}$



	A (m)	B (m)	C (m)	D (m)	E (m)
H3DMAP	1.9423	2.0101	0.4182	0.1000	0.1171
Experiment	2.00	2.066	0.4328	0.1328	0.1172

FIGURE 2.68.4

CASE 68: COMPARISON OF FINAL DEFORMATION BETWEEN H3DMAP
AND EXPERIMENTAL VALUES

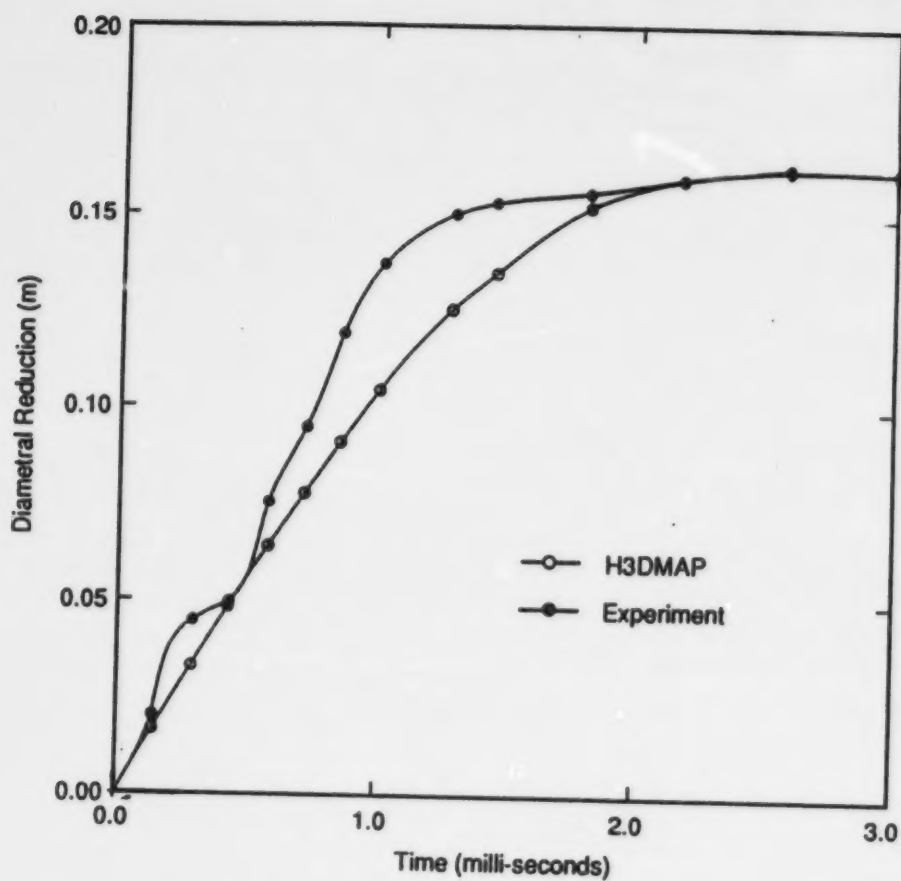


FIGURE 2.68.5

CASE 68: TIME HISTORY PLOT OF DIAMETRAL
REDUCTION FOR THE SPHERE IMPACT ANALYSIS
(Comparison between H3DMAP and Experimental Values)

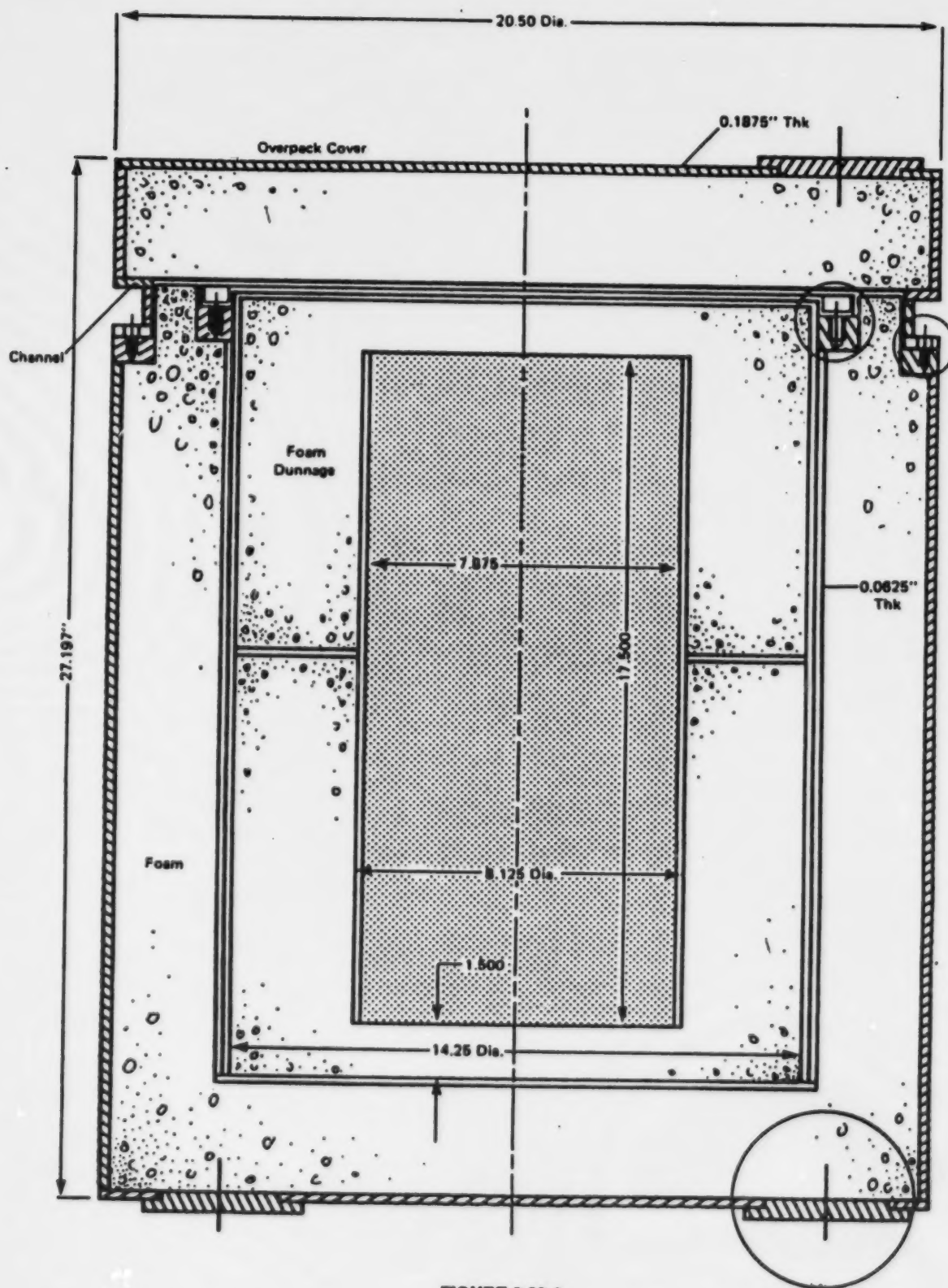


FIGURE 2.69.1

CASE 69: SCALE MODEL RFTP ASSEMBLY

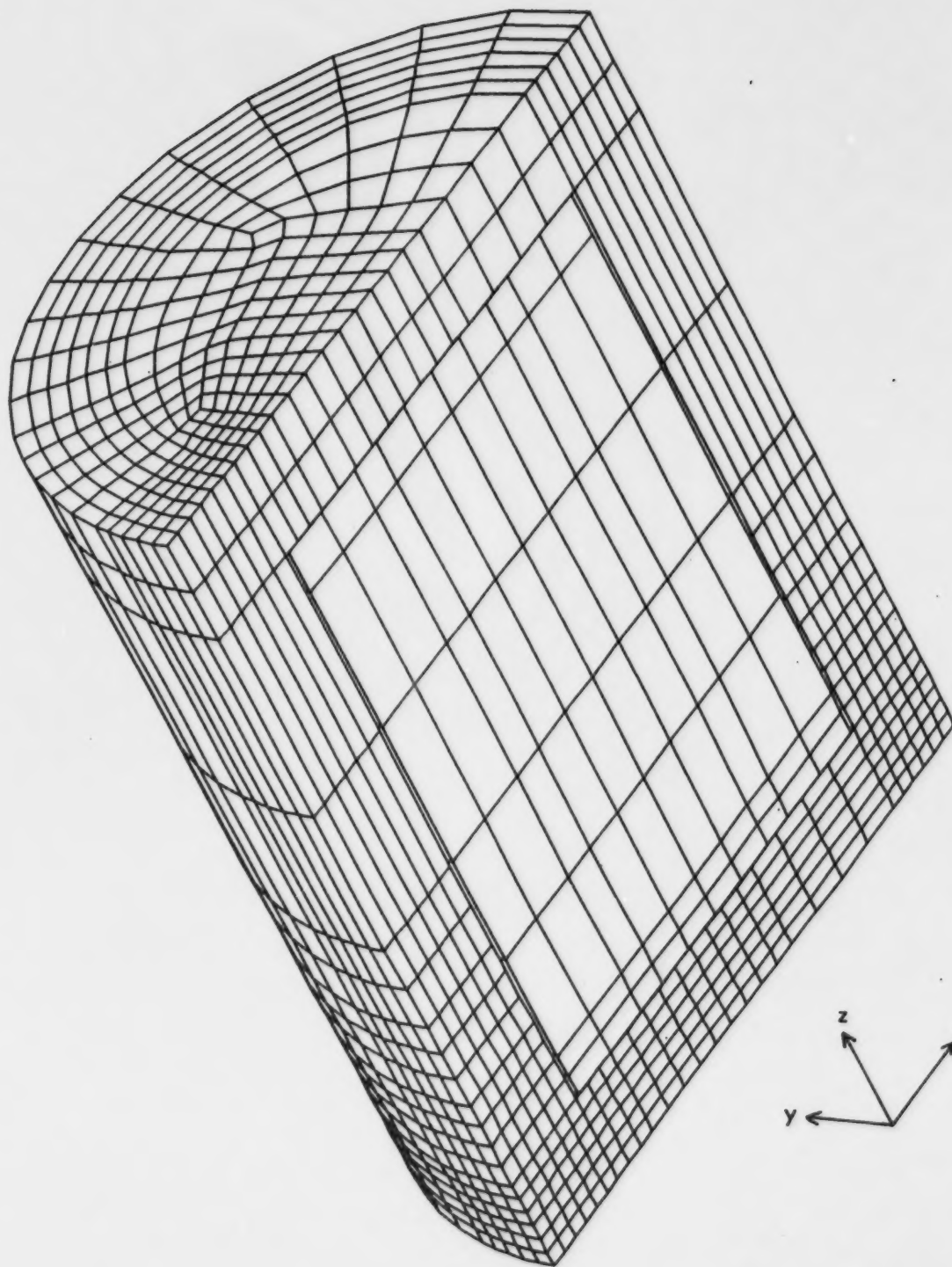


FIGURE 2.88.2

CASE 69: RFTP COMPUTER MODEL

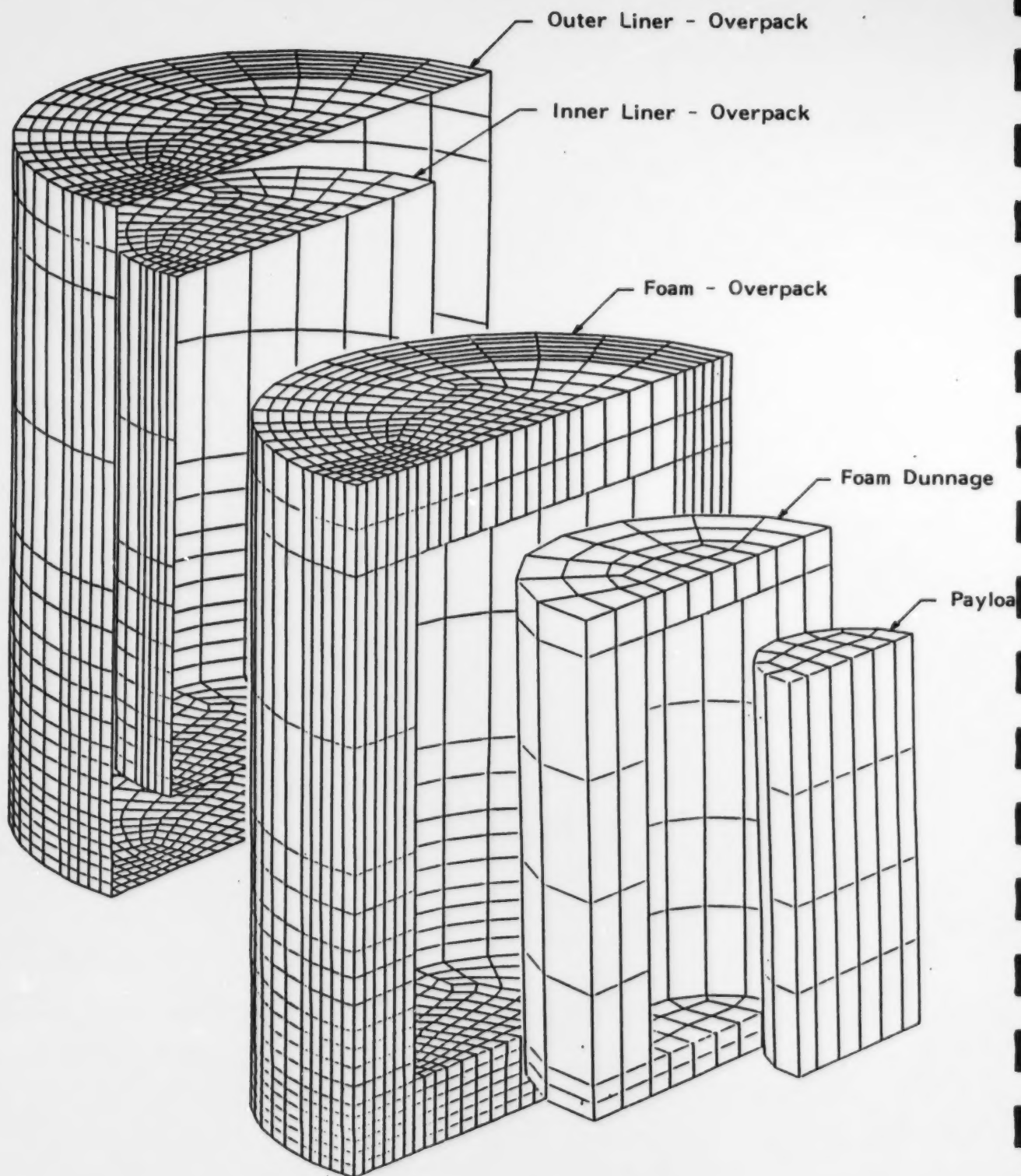


FIGURE 2.69.3

CASE 69: EXPLODED VIEW OF THE RFTP COMPUTER MODEL

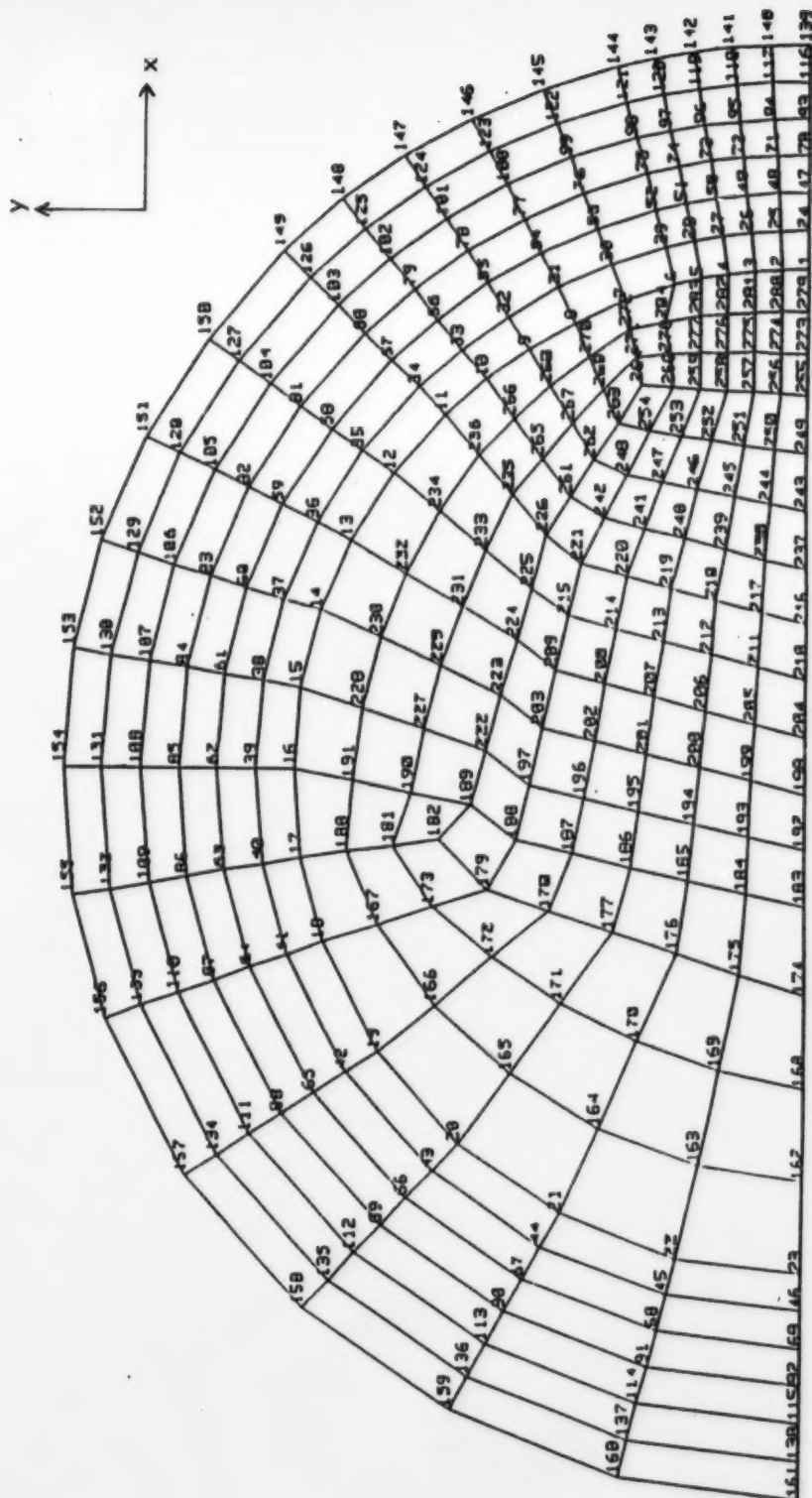


FIGURE 2.69.4
CASE 69: TEMPLATE FOR THE RFTP COMPUTER MODEL



FIGURE 2.69.5

CASE 69: RFTP 9 m DROP TEST SET-UP

89-221
185397-B30-RD

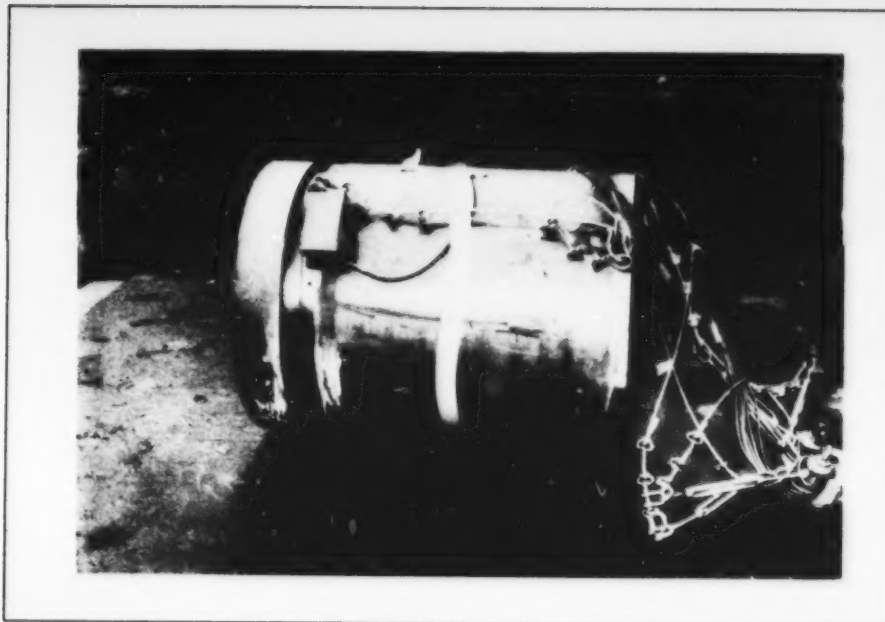


FIGURE 2.69.6

CASE 69: RFTP AFTER 9 m CORNER DROP - EXTERNAL

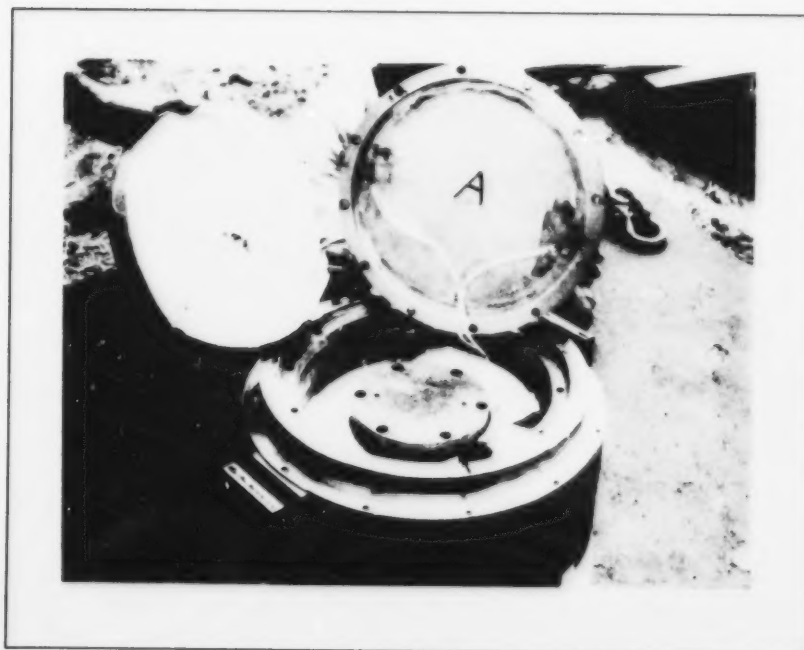


FIGURE 2.69.7

CASE 69: RFTP AFTER 9 m CORNER DROP - INTERNAL



FIGURE 2.69.8

**CASE 69: EXPERIMENTAL RFTP MODEL
POST-IMPACT DEFORMATION**

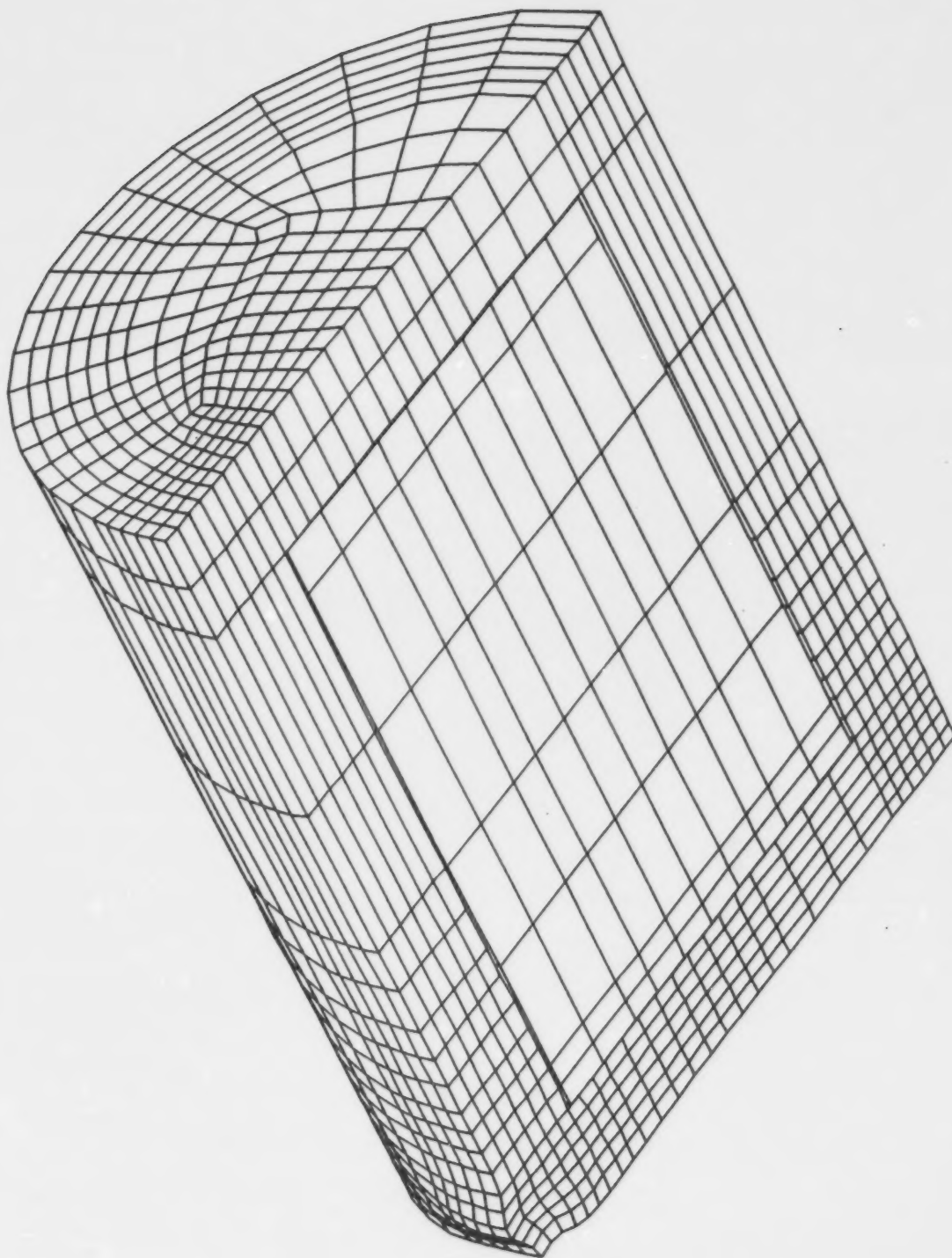


FIGURE 2.69.9

CASE 69: DEFORMED GEOMETRY AT 1 ms

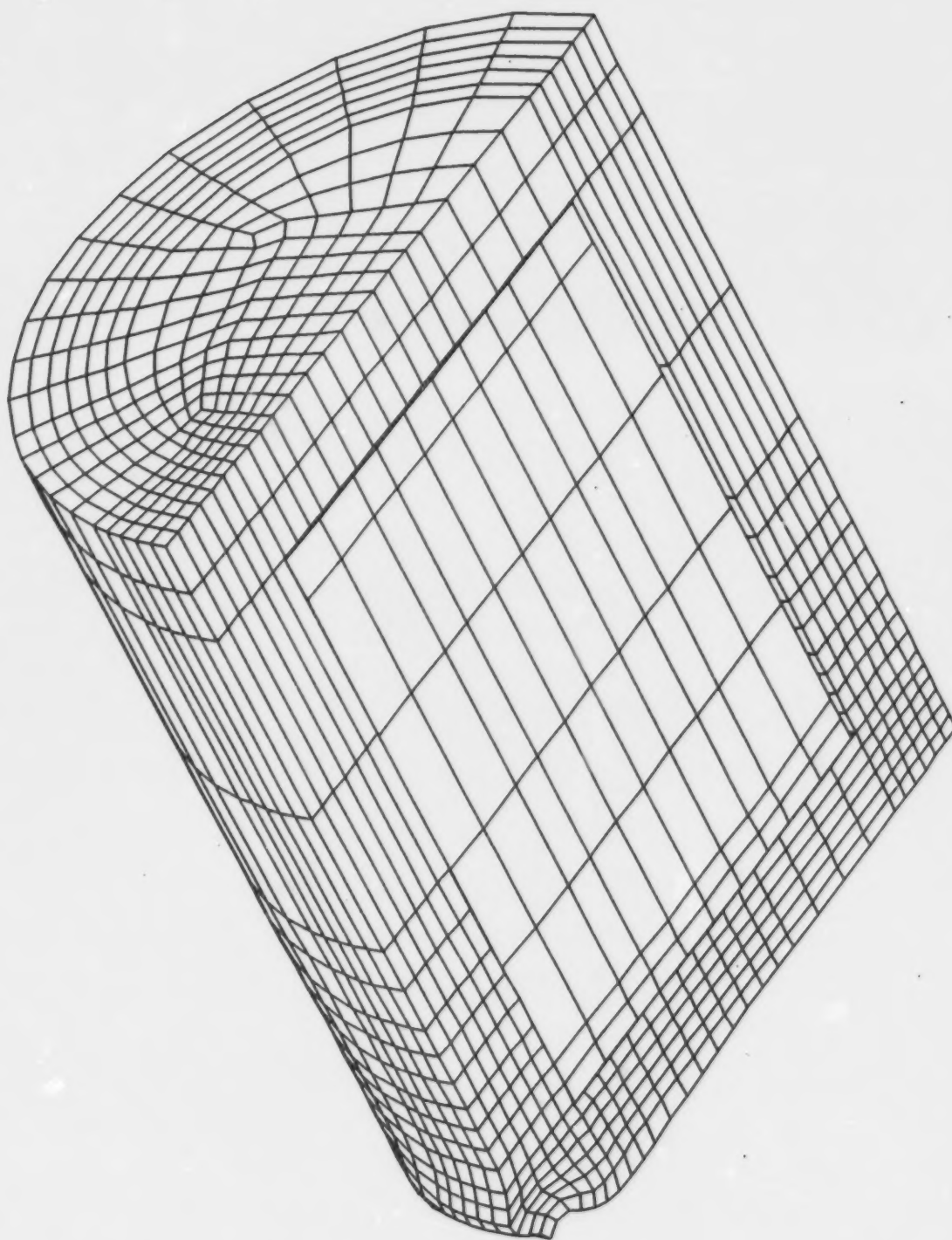


FIGURE 2.69.10

CASE 69: DEFORMED GEOMETRY AT 2 ms

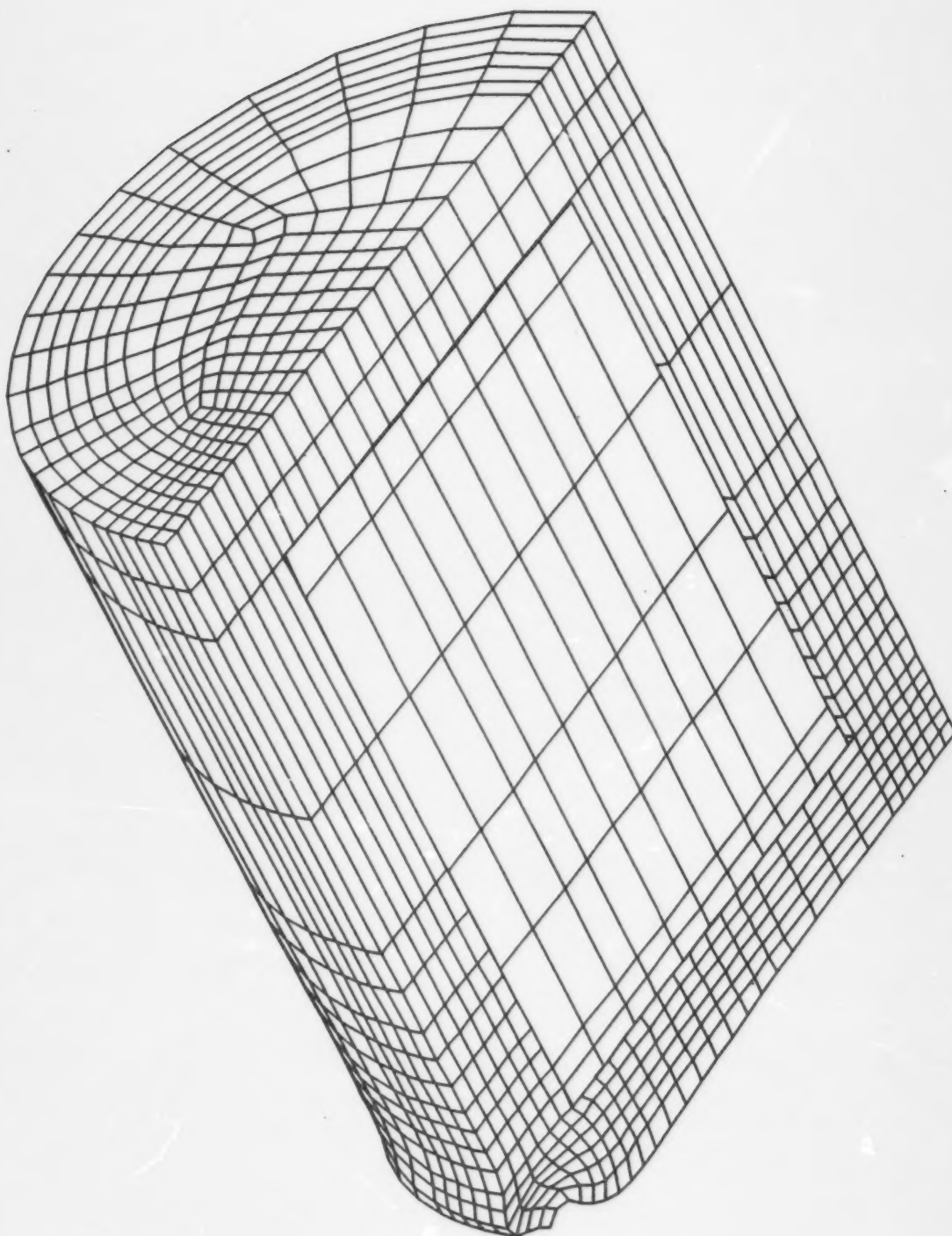


FIGURE 2.69.11

CASE 69: DEFORMED GEOMETRY AT 3 ms

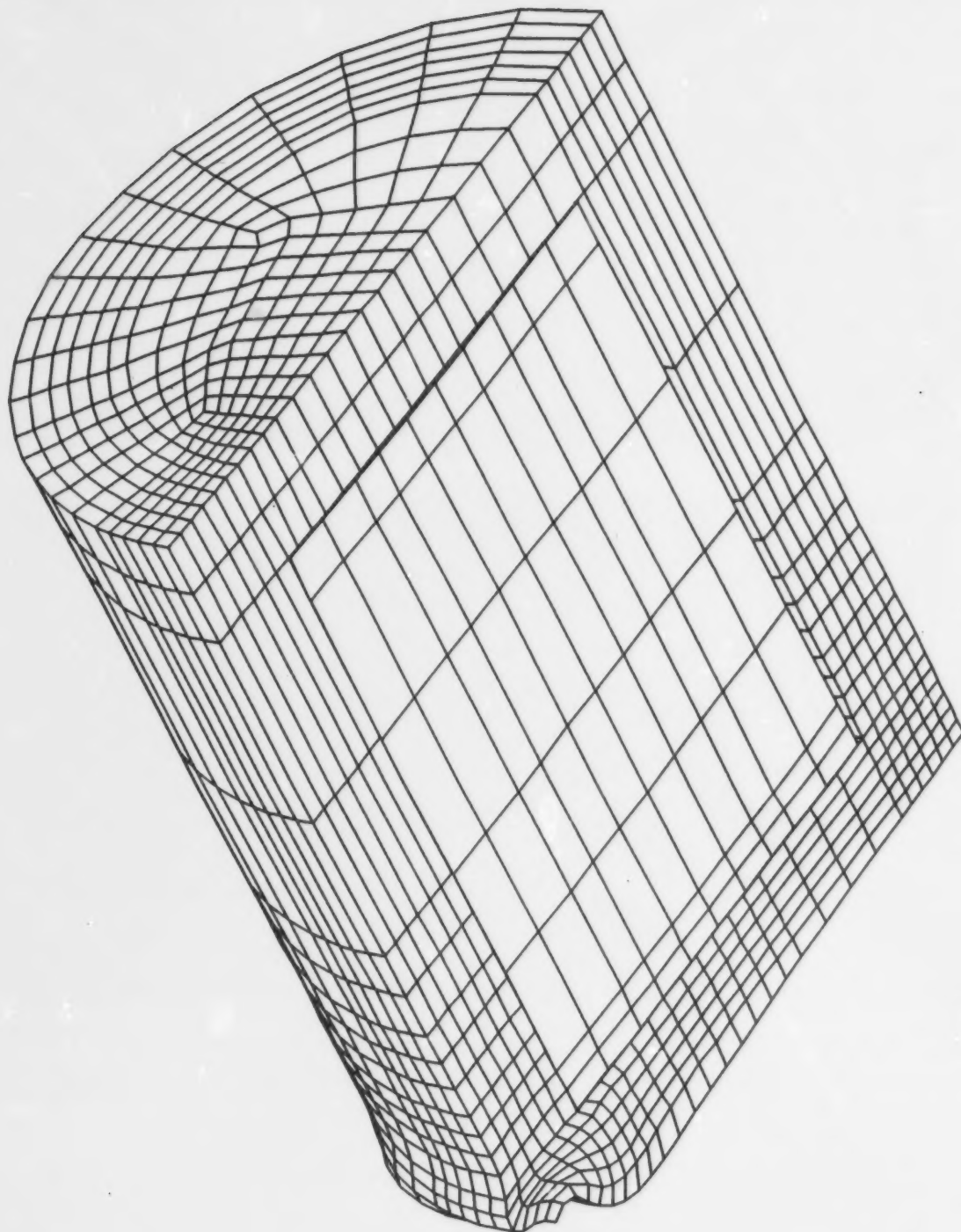


FIGURE 2.69.12

CASE 69: DEFORMED GEOMETRY AT 4 ms

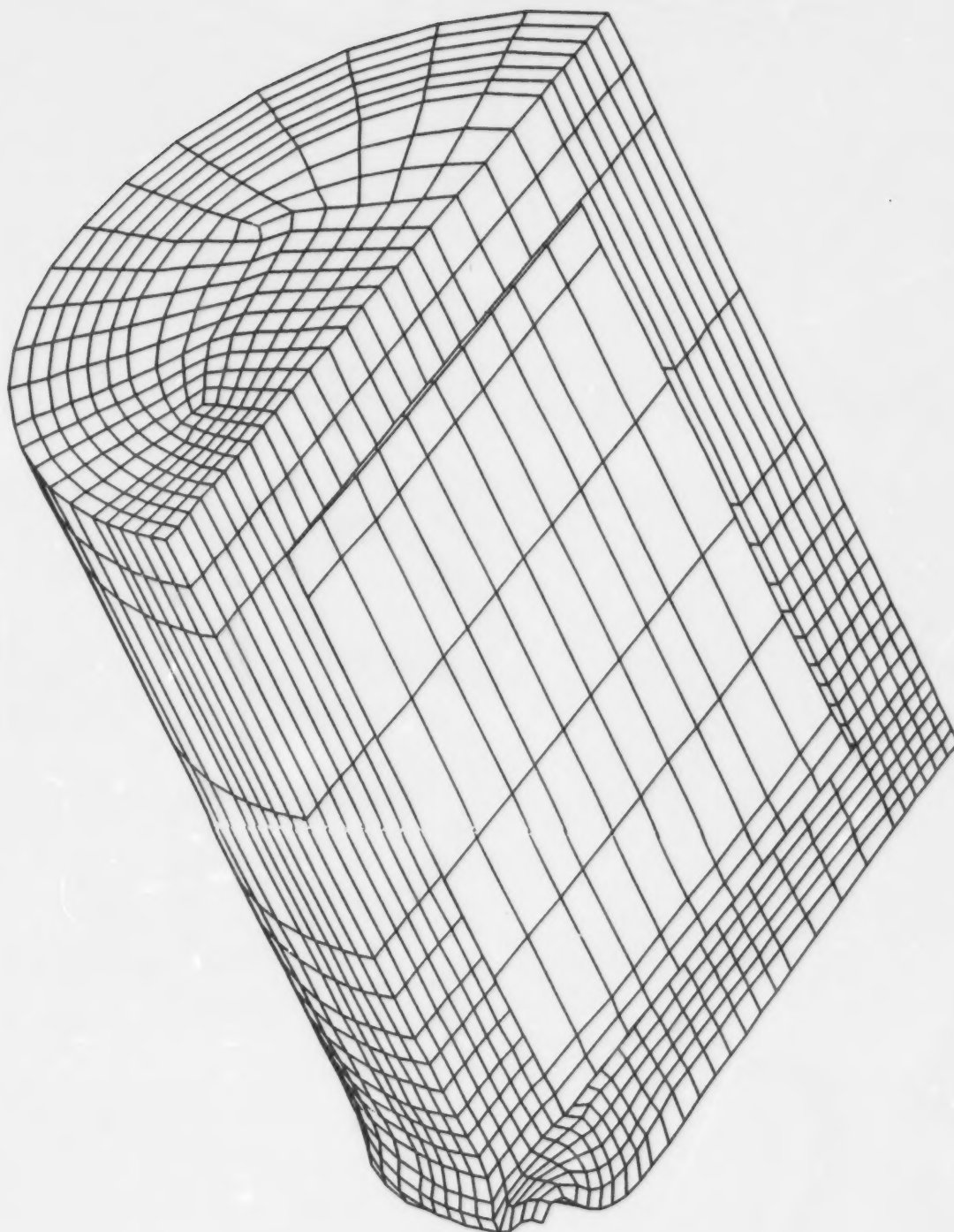


FIGURE 2.69.13

CASE 69: DEFORMED GEOMETRY AT 5 ms

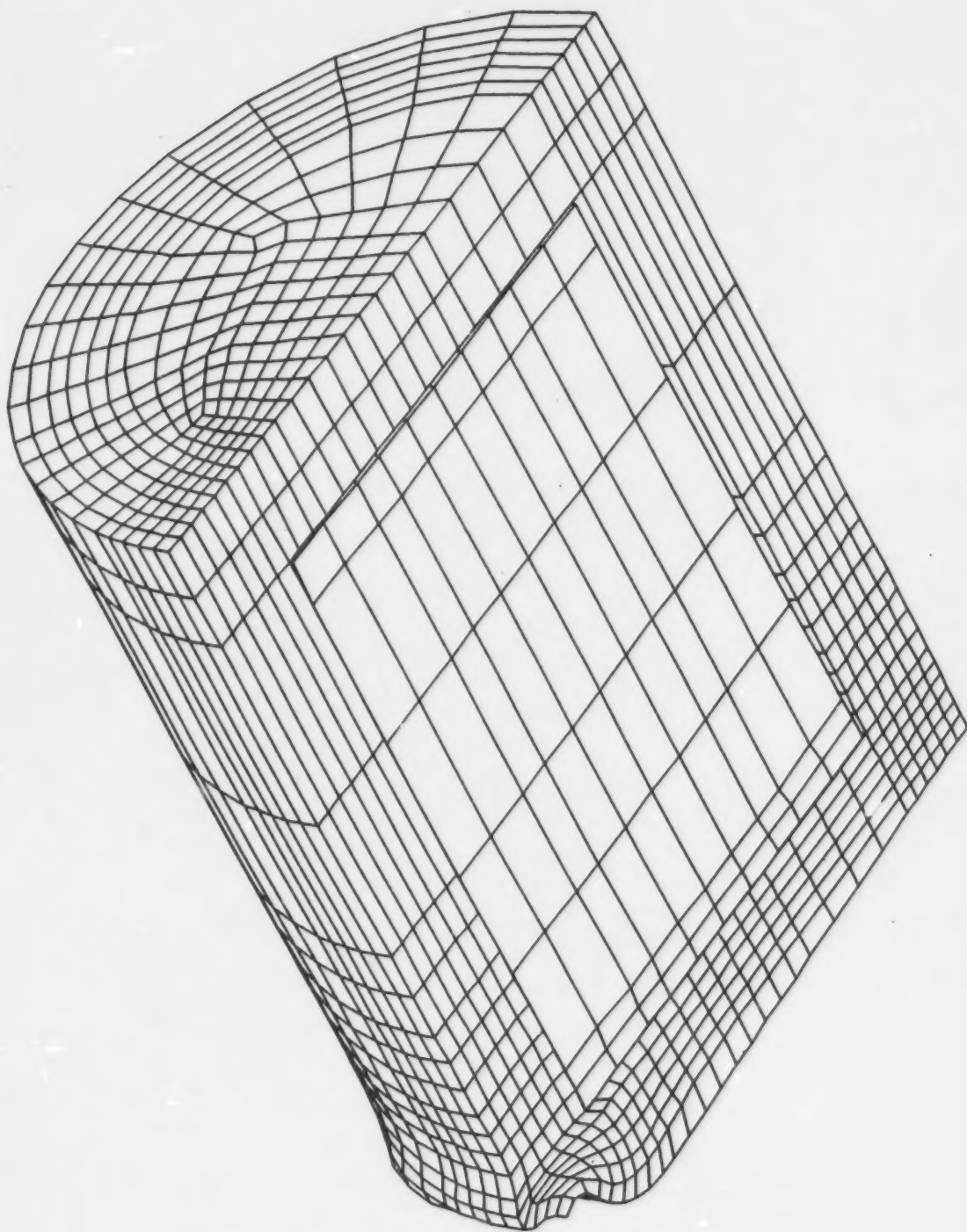


FIGURE 2.69.14

CASE 69: DEFORMED GEOMETRY AT 6 ms

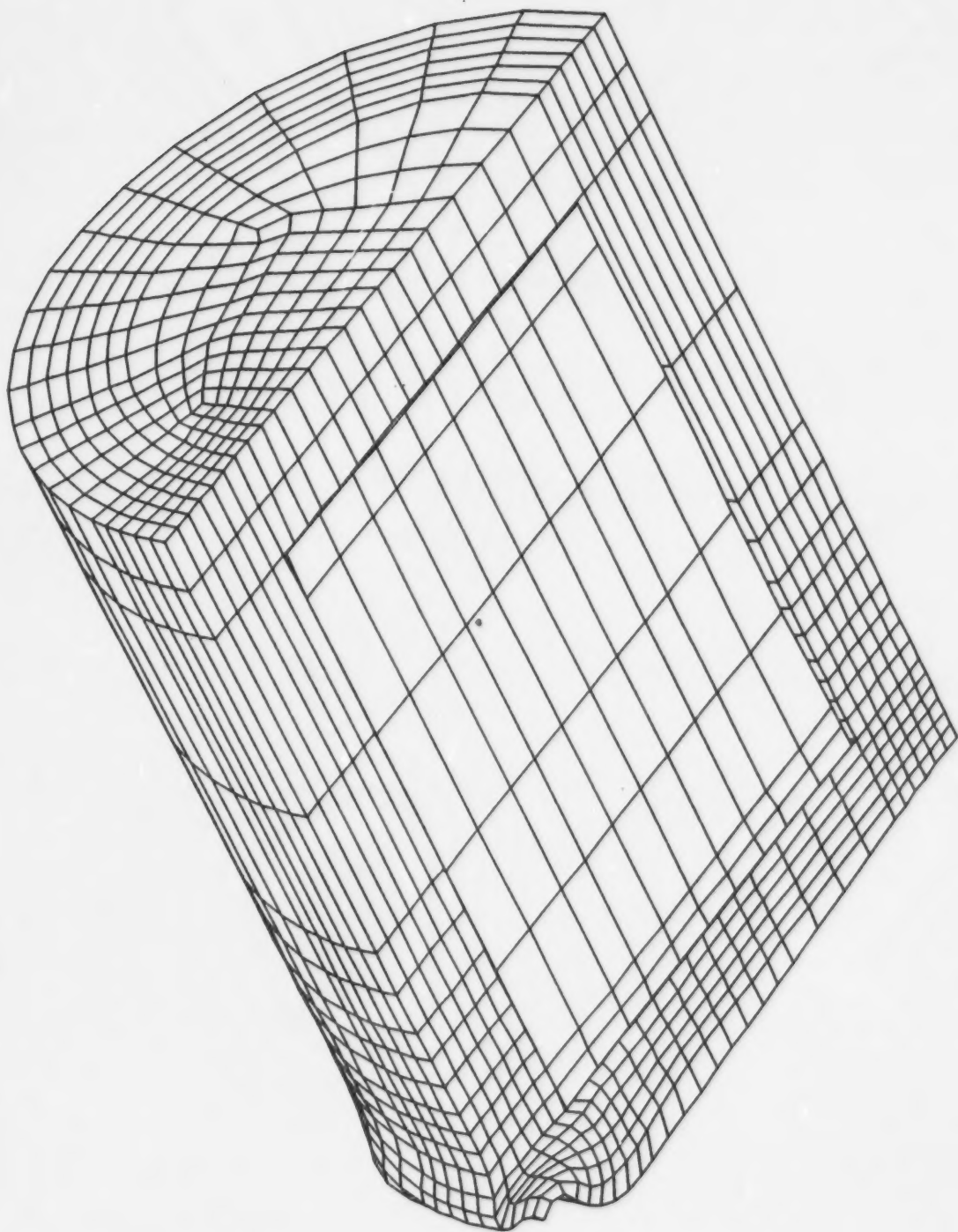


FIGURE 2.69.15

CASE 69: DEFORMED GEOMETRY AT 7 ms

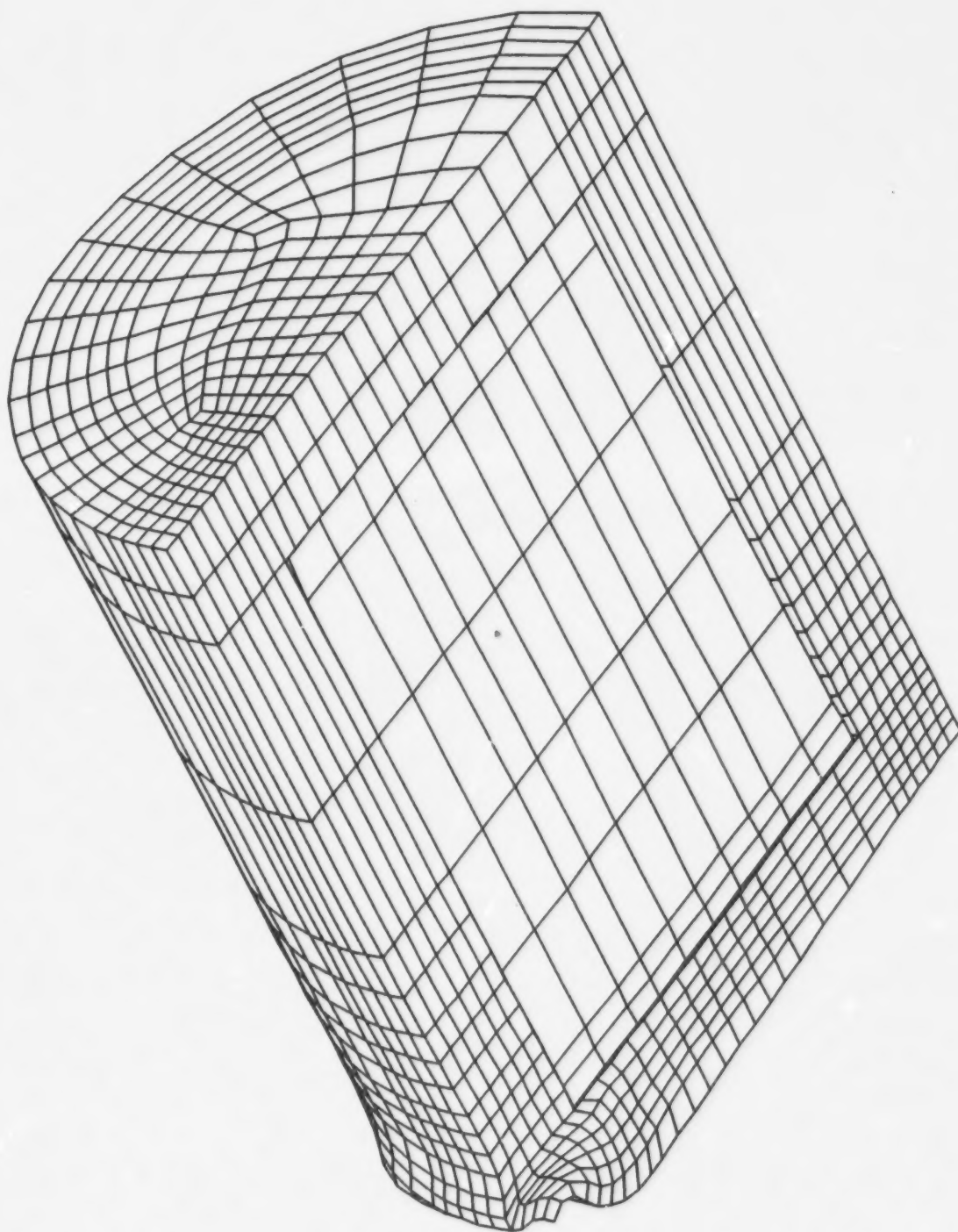


FIGURE 2.69.16

CASE 69: DEFORMED GEOMETRY AT 8 ms

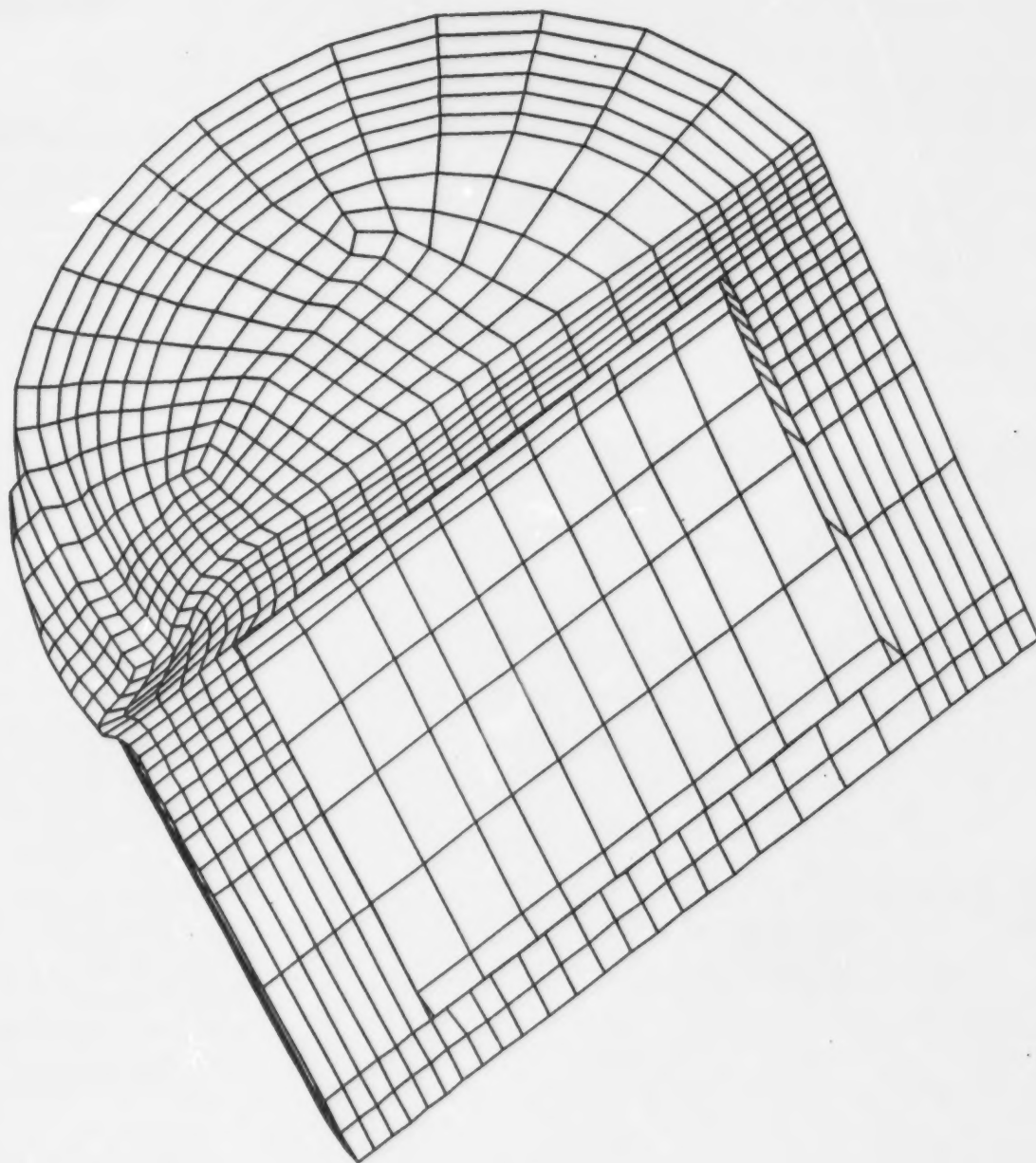


FIGURE 2.69.17
CASE 69: DEFORMED GEOMETRY AT 8 ms - VIEW A

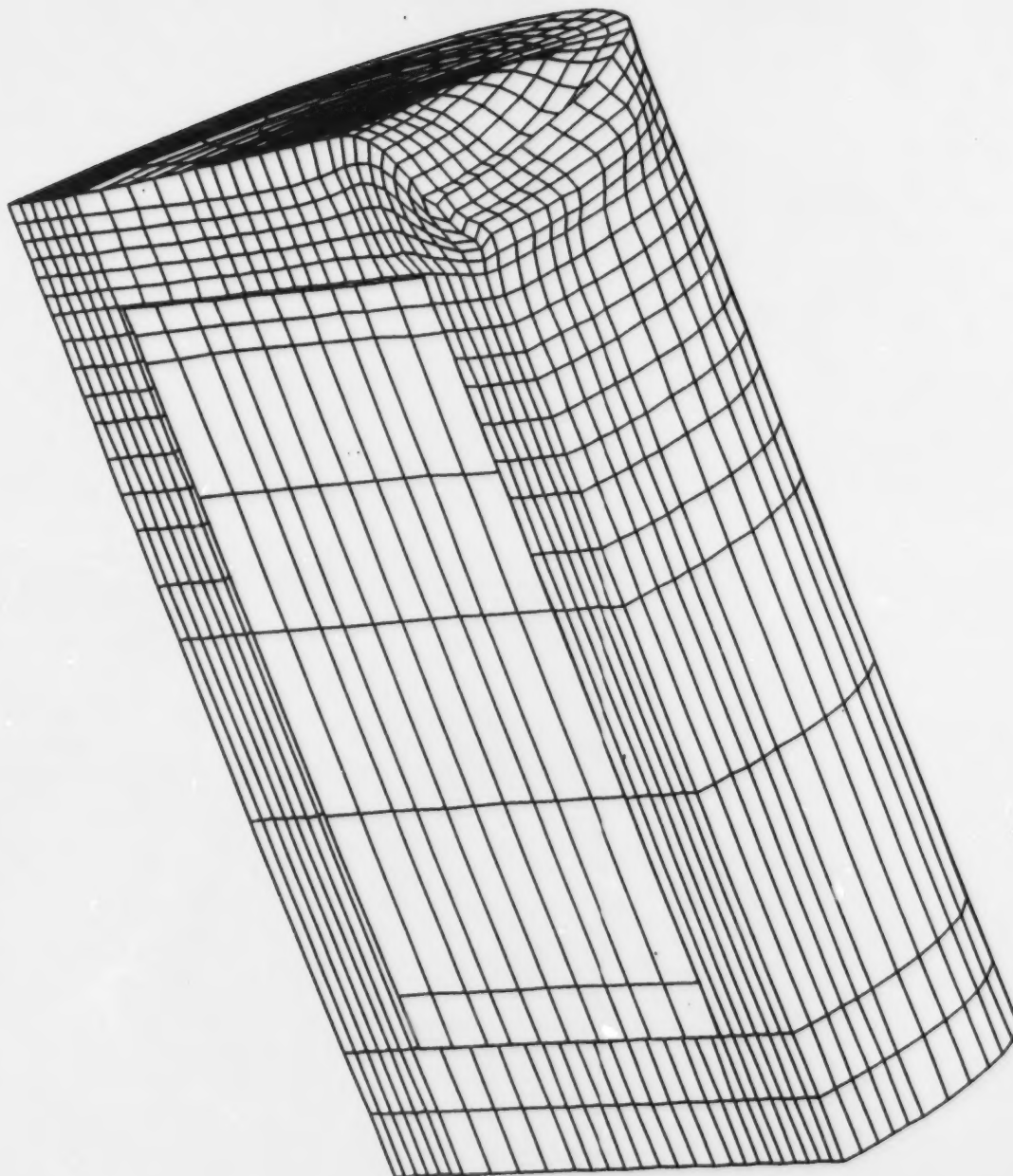


FIGURE 2.69.18

CASE 69: DEFORMED GEOMETRY AT 8 ms - VIEW B

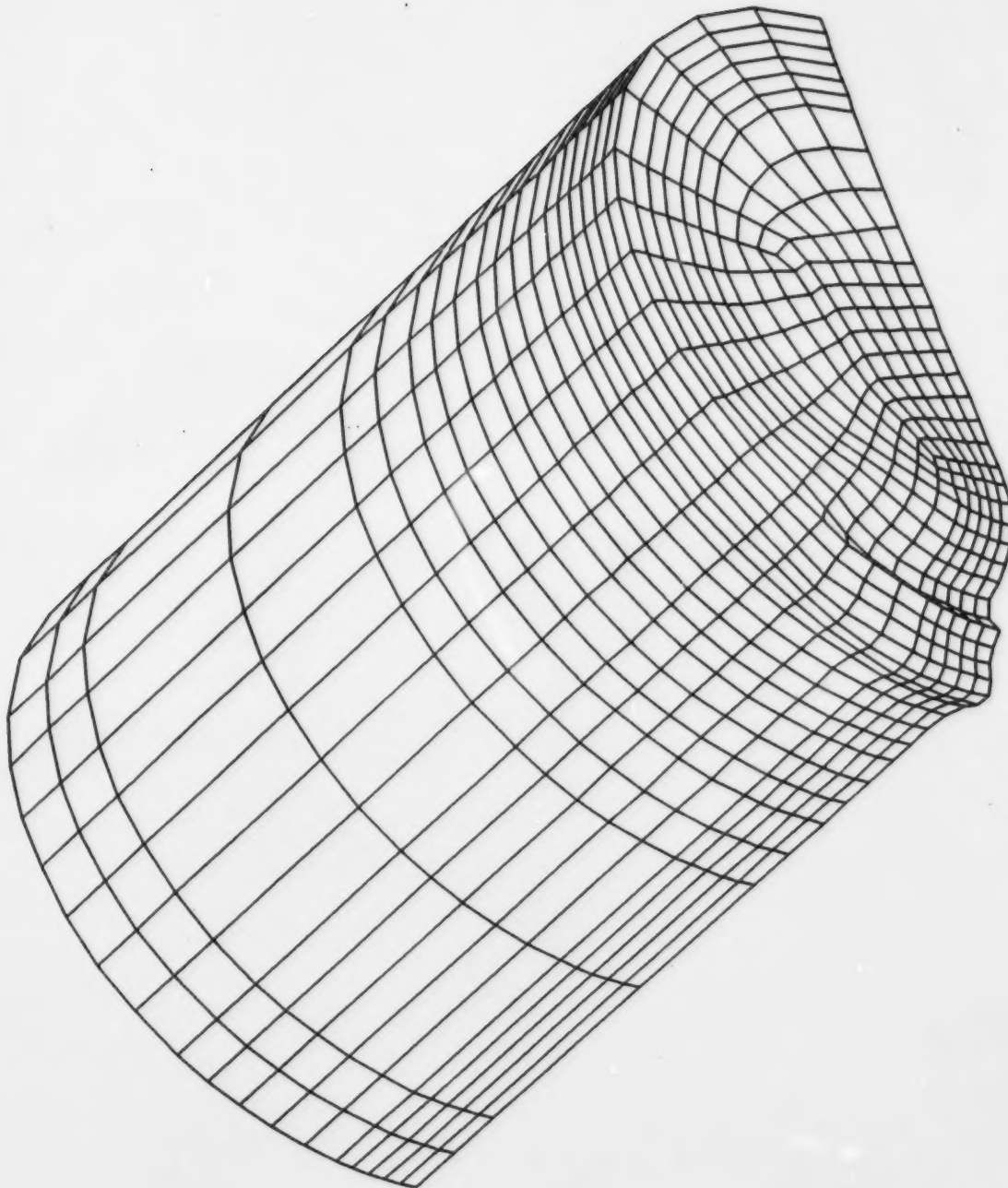


FIGURE 2.69.19

CASE 69: DEFORMED GEOMETRY AT 8 ms - VIEW C

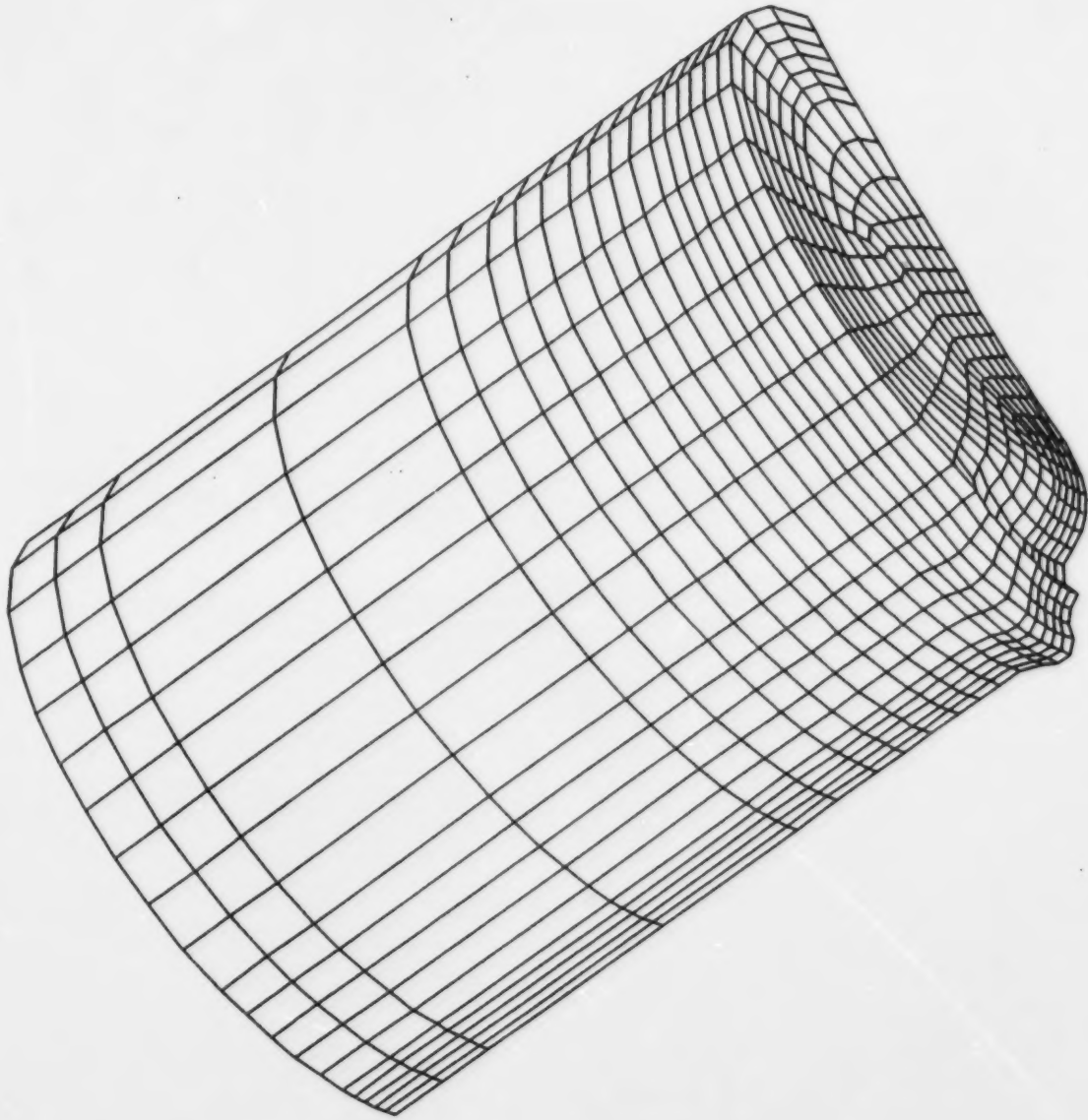


FIGURE 2.69.20

CASE 69: DEFORMED GEOMETRY AT 8 ms - VIEW D

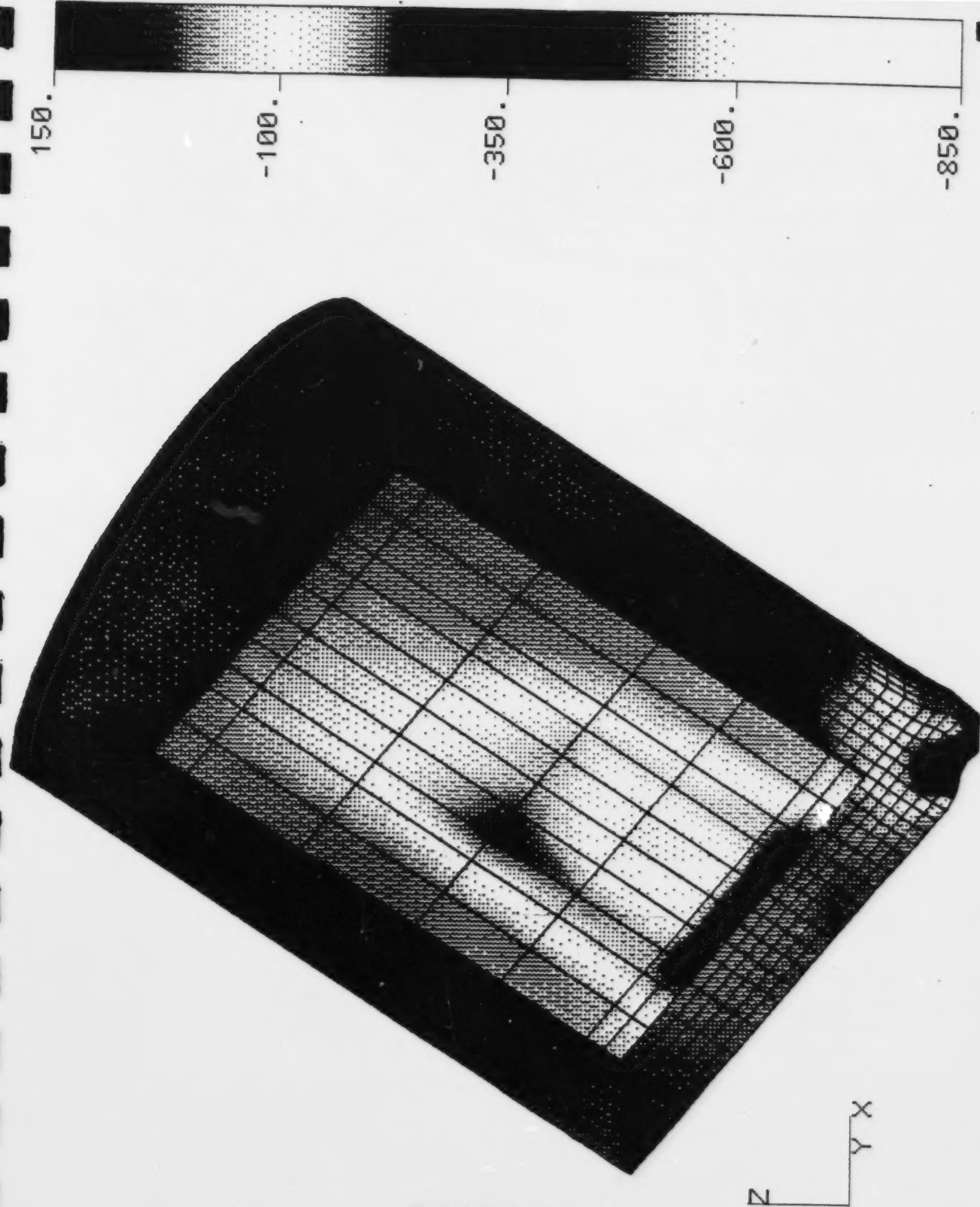


FIGURE 2.69.21
CASE 69: HYDRODYNAMIC STRESS DISTRIBUTION
AT TIME = 1 ms

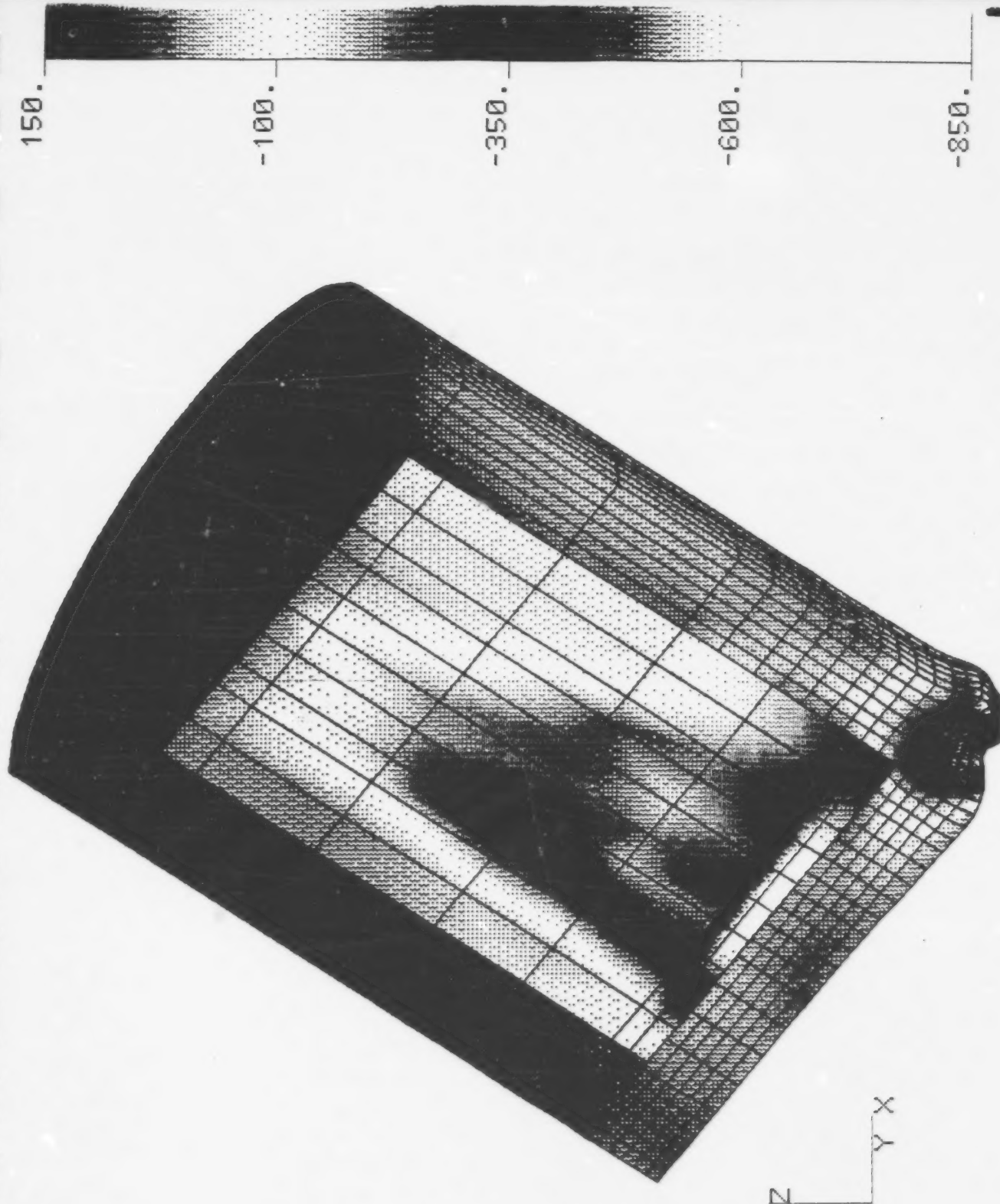


FIGURE 2.69.22
CASE 69: HYDRODYNAMIC STRESS DISTRIBUTION
AT TIME = 2 ms

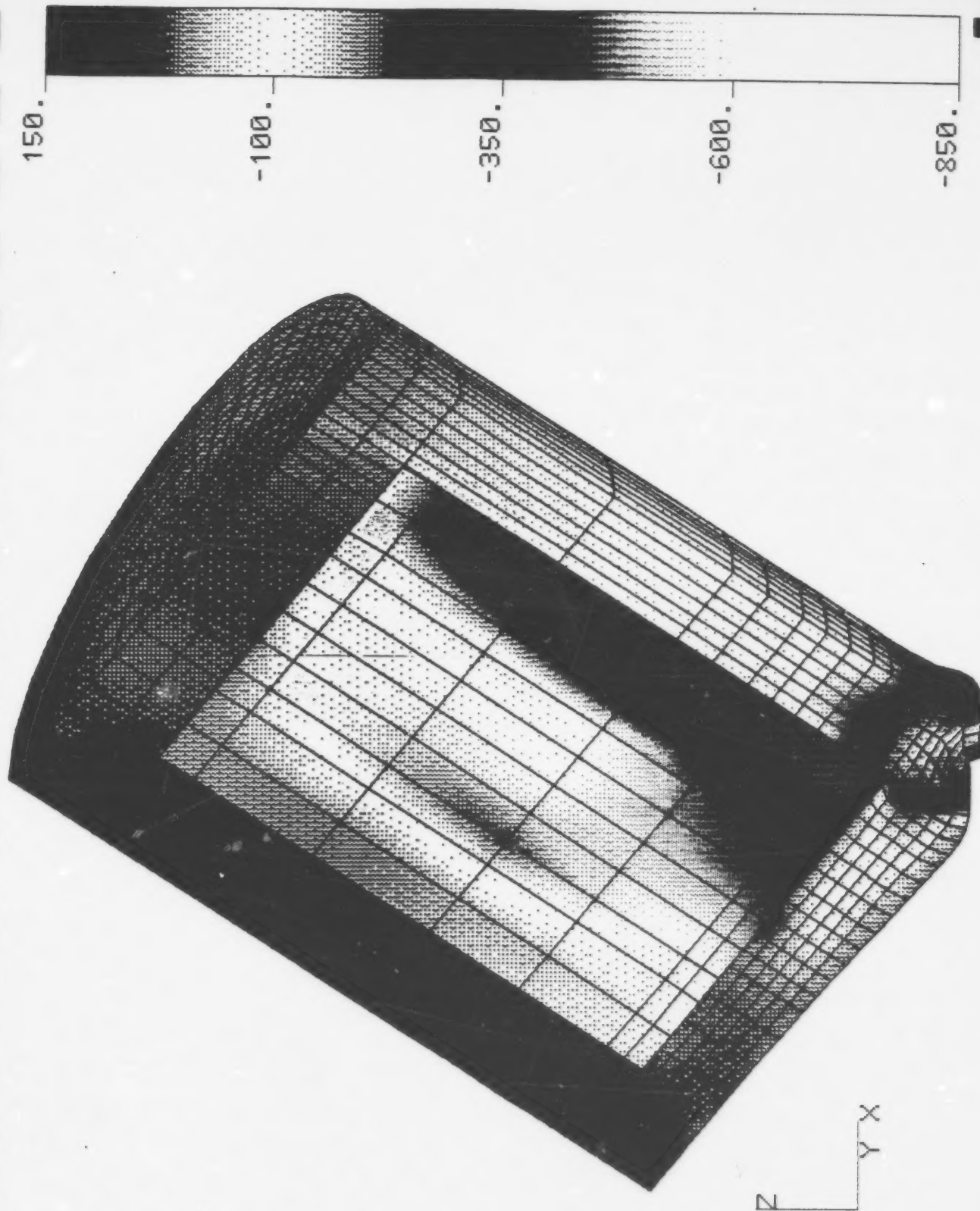


FIGURE 2.69.23
CASE 69: HYDRODYNAMIC STRESS DISTRIBUTION
AT TIME = 3 ms

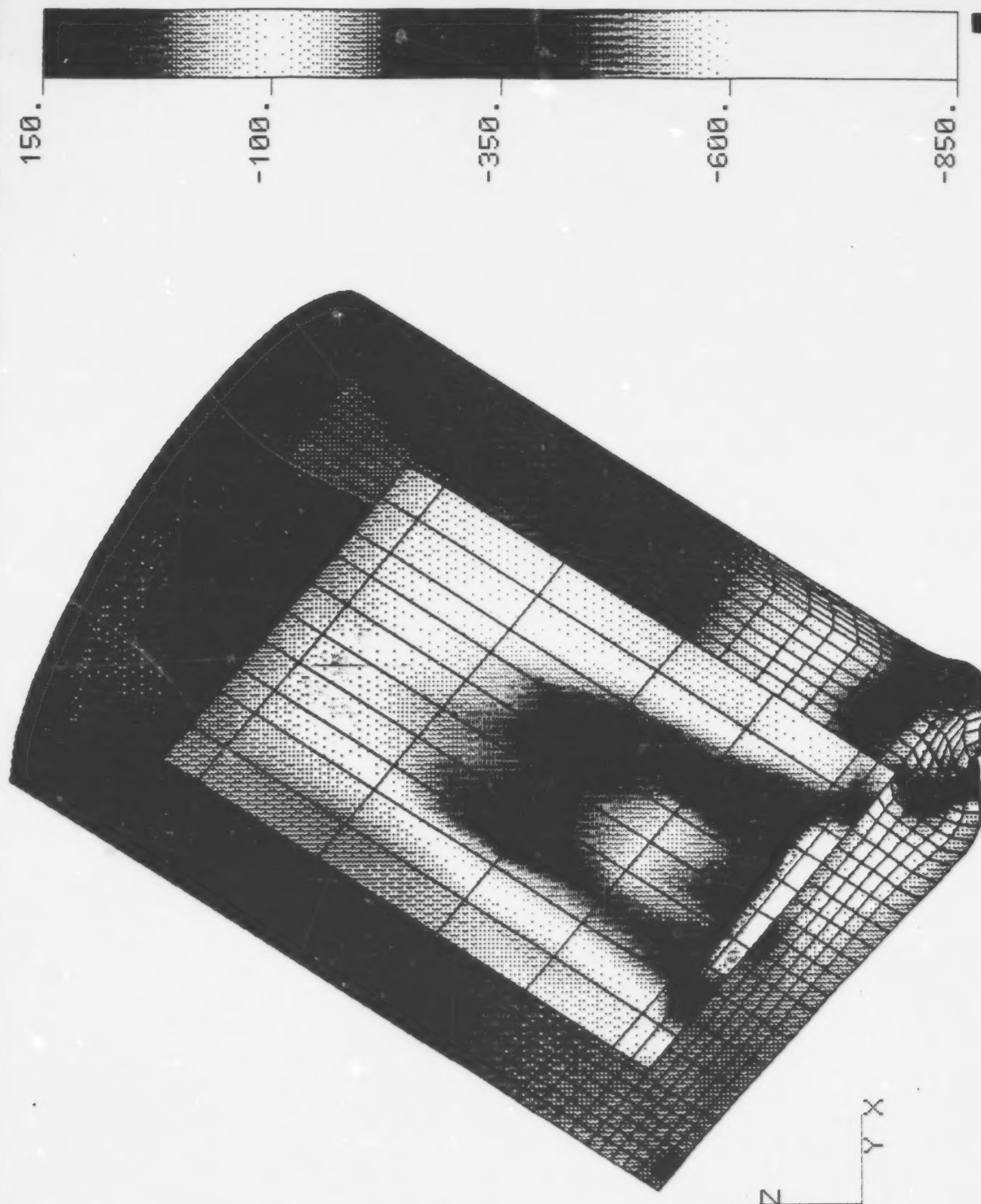


FIGURE 2.69.24
CASE 69: HYDRODYNAMIC STRESS DISTRIBUTION
AT TIME = 4 ms

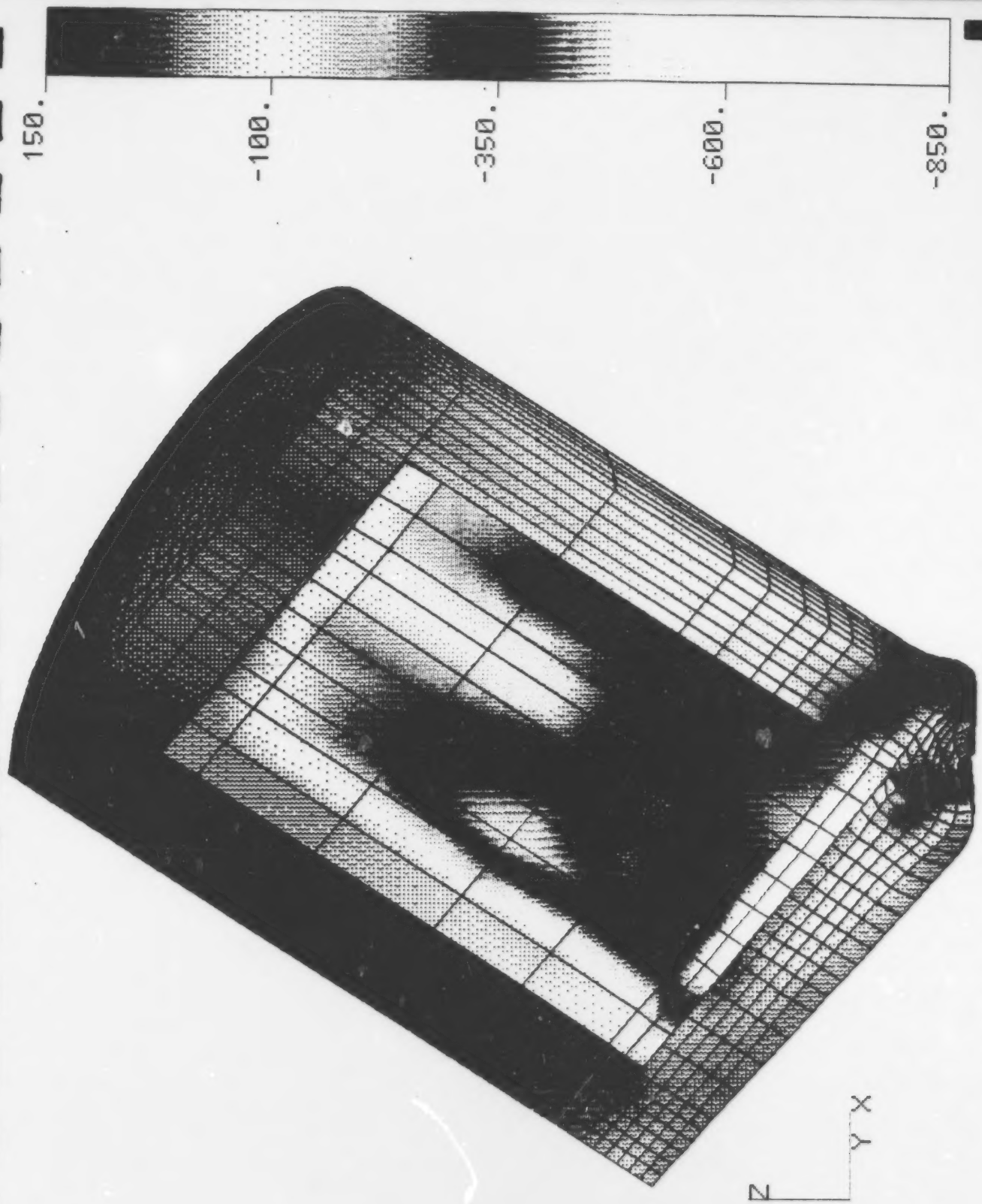


FIGURE 2.69.25
CASE 69: HYDRODYNAMIC STRESS DISTRIBUTION
AT TIME = 5 ms

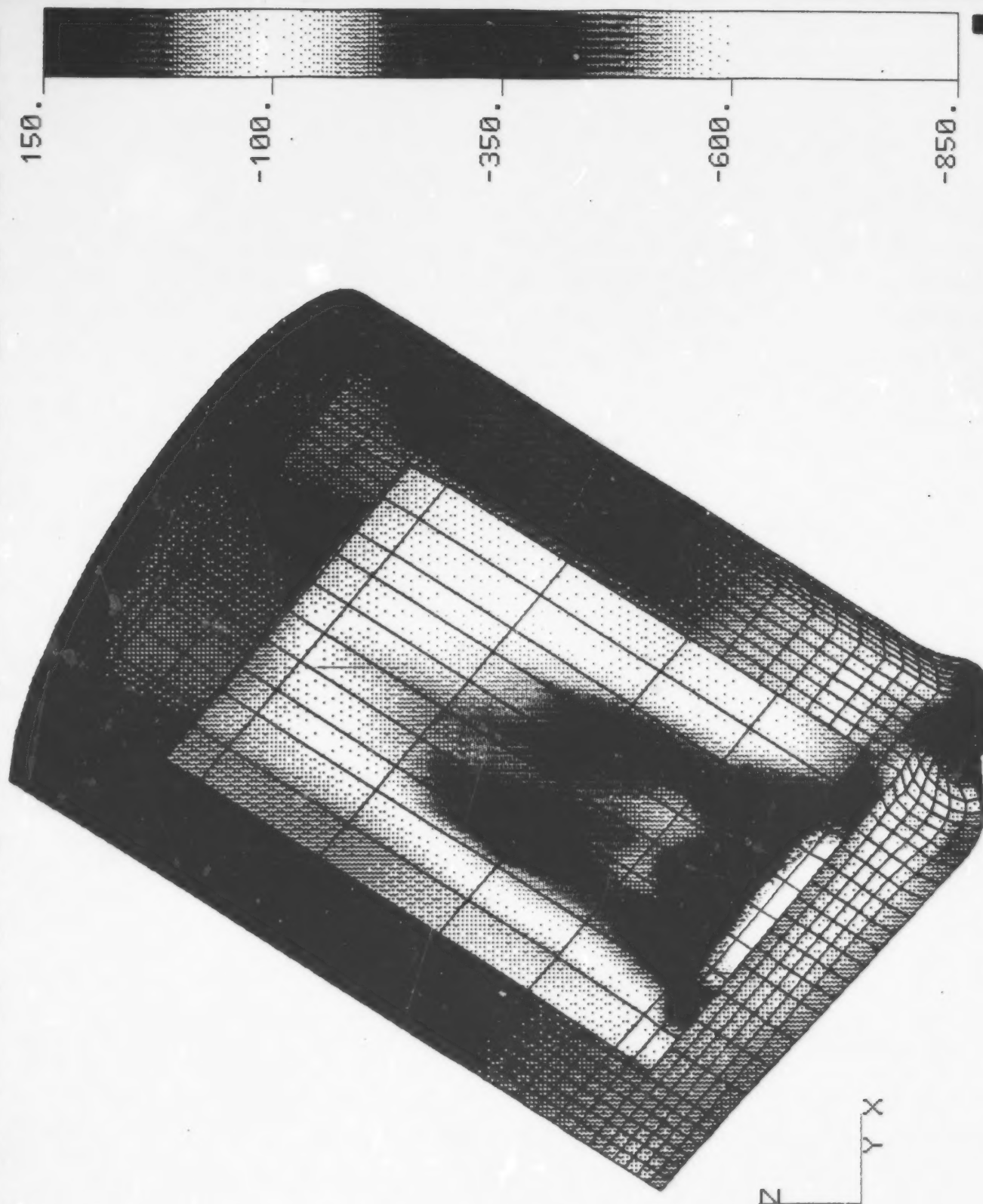


FIGURE 2.69.26
CASE 69: HYDRODYNAMIC STRESS DISTRIBUTION
AT TIME = 6 ms

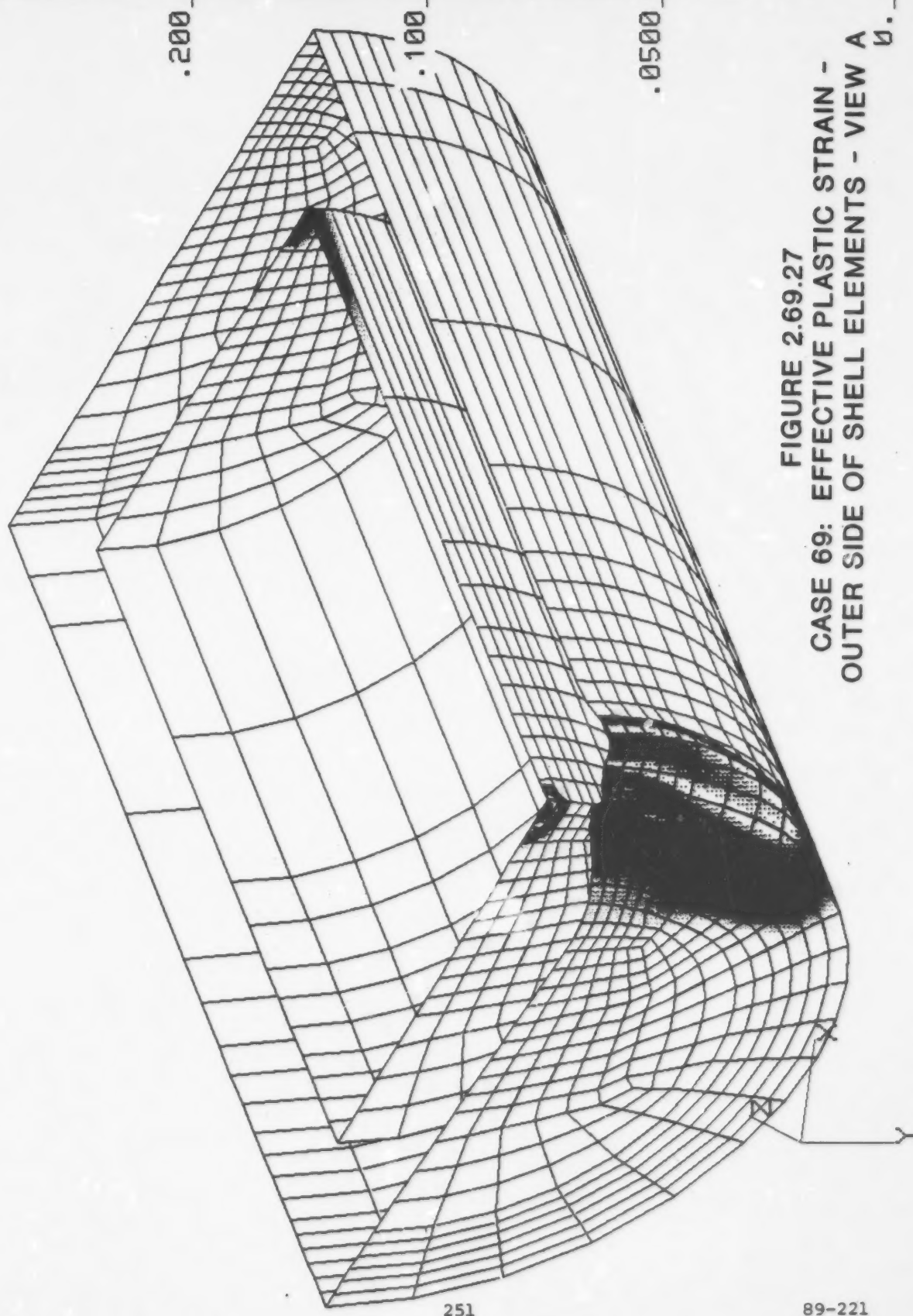


FIGURE 2.69.27
CASE 69: EFFECTIVE PLASTIC STRAIN -
OUTER SIDE OF SHELL ELEMENTS - VIEW A

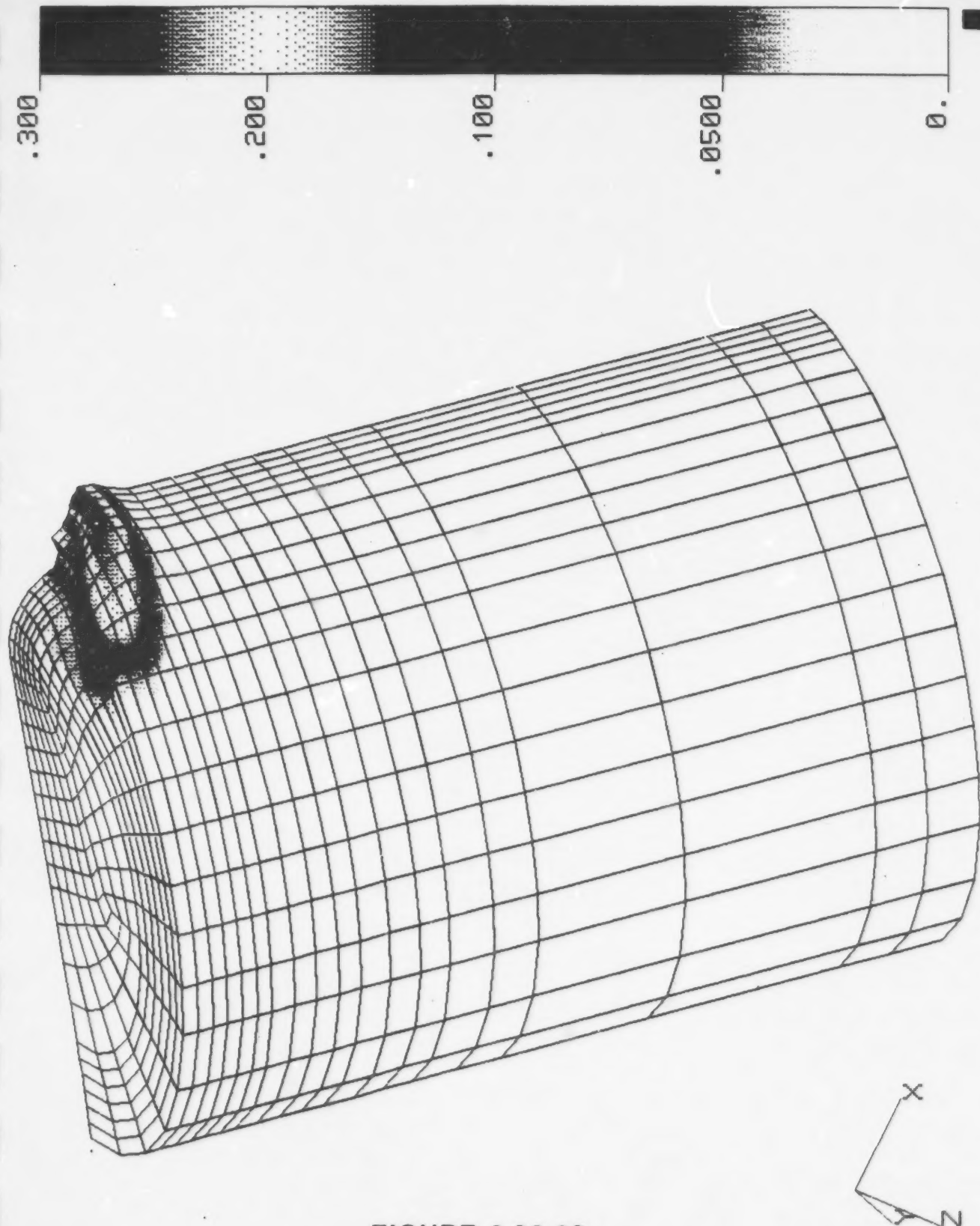


FIGURE 2.69.28
CASE 69: EFFECTIVE PLASTIC STRAIN -
OUTER SIDE OF SHELL ELEMENTS - VIEW B

.300

.200

.100

.0500

0.

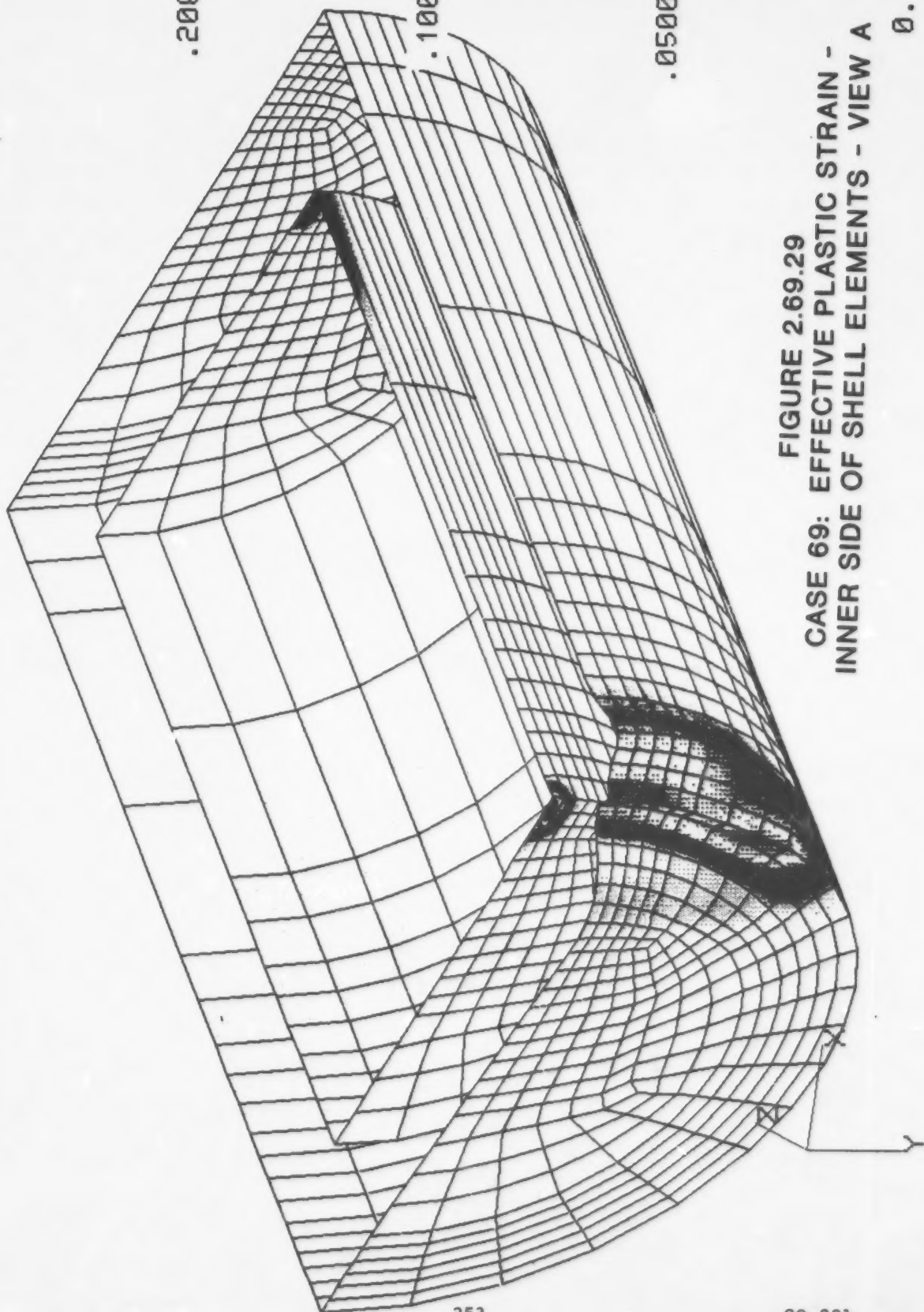


FIGURE 2.69.29
CASE 69: EFFECTIVE PLASTIC STRAIN -
INNER SIDE OF SHELL ELEMENTS - VIEW A

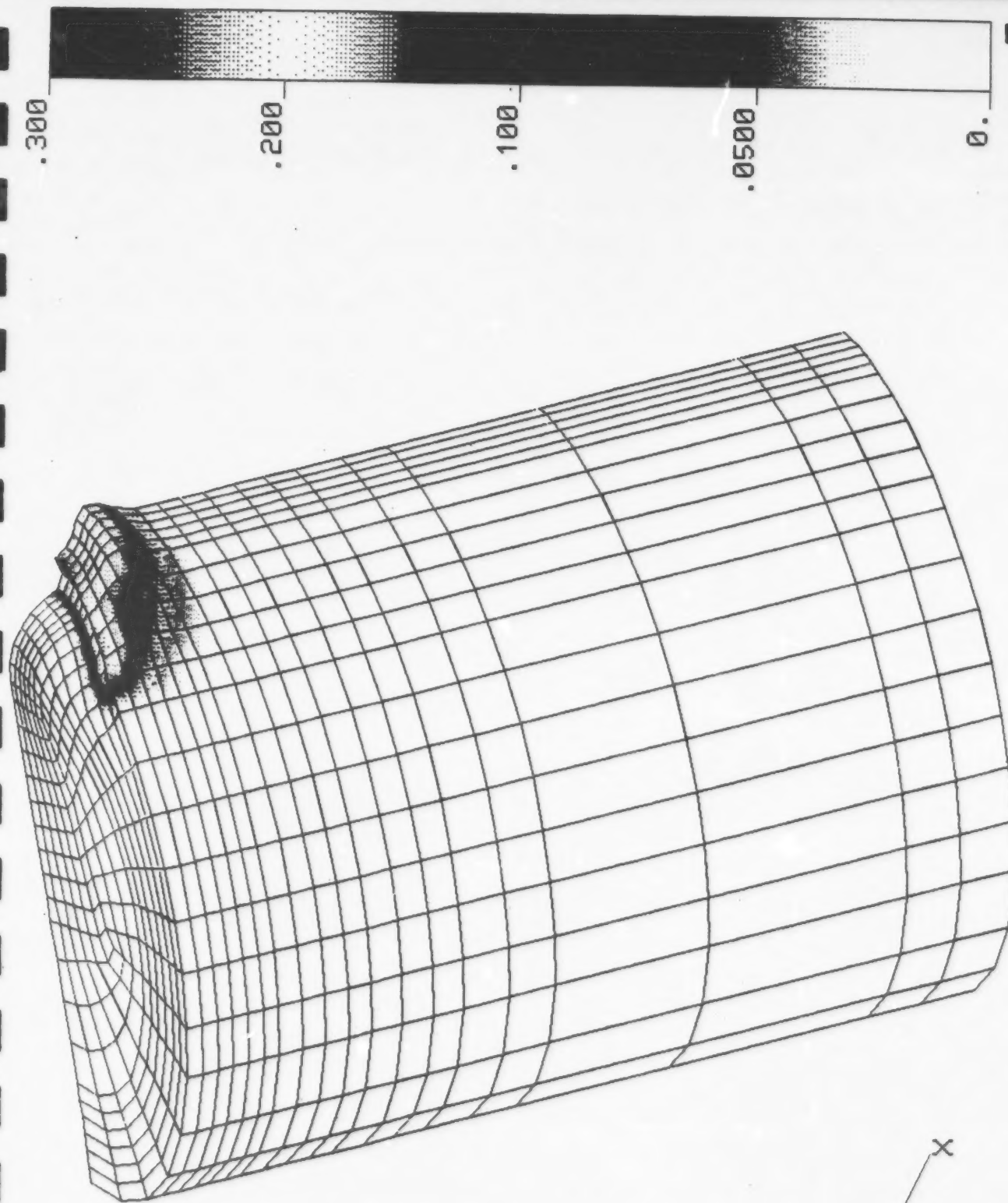


FIGURE 2.69.30
CASE 69: EFFECTIVE PLASTIC STRAIN -
INNER SIDE OF SHELL ELEMENTS - VIEW B

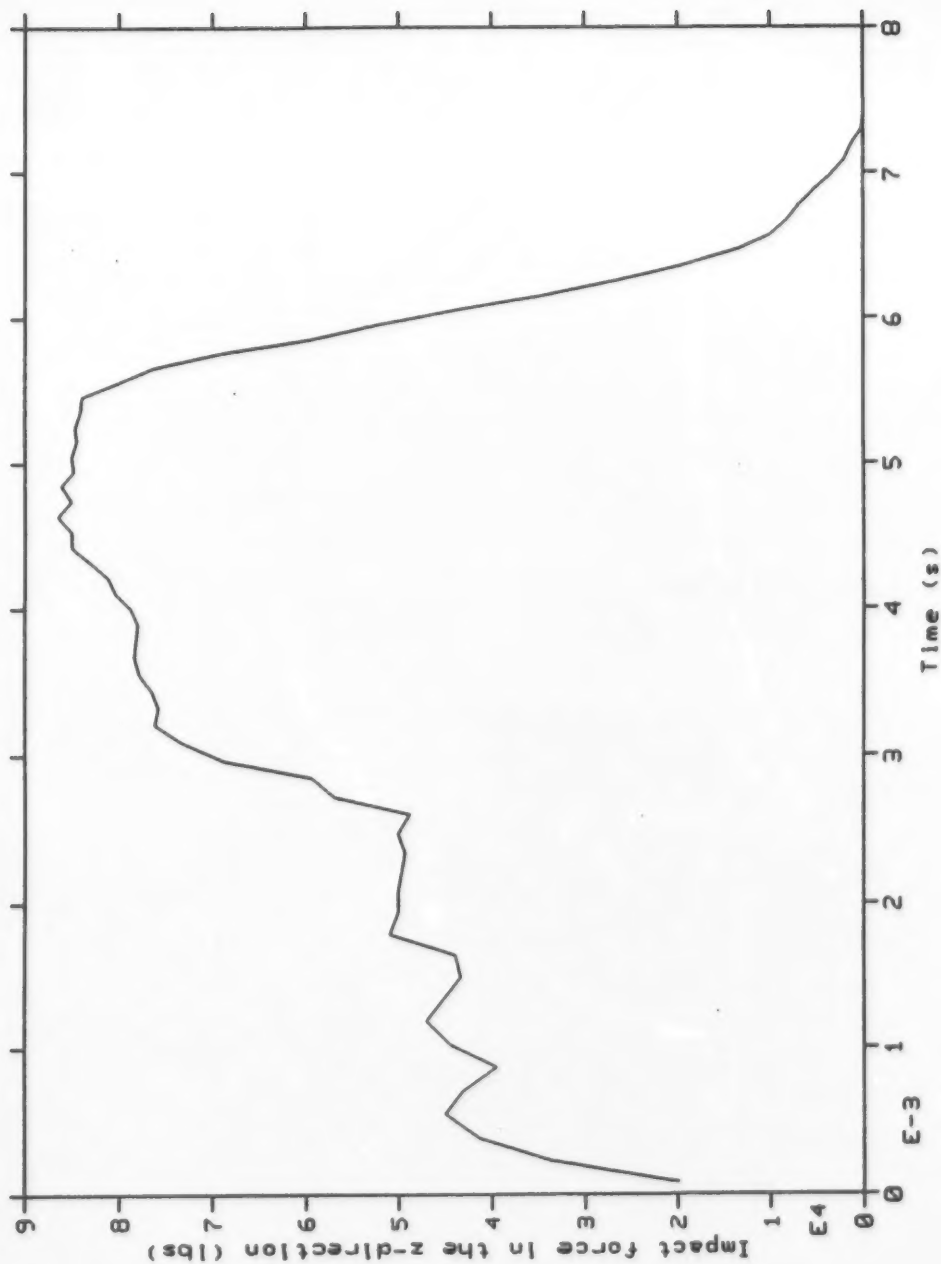


FIGURE 2.60.31

CASE 60: TIME HISTORY OF IMPACT FORCE IN THE Z-DIRECTION

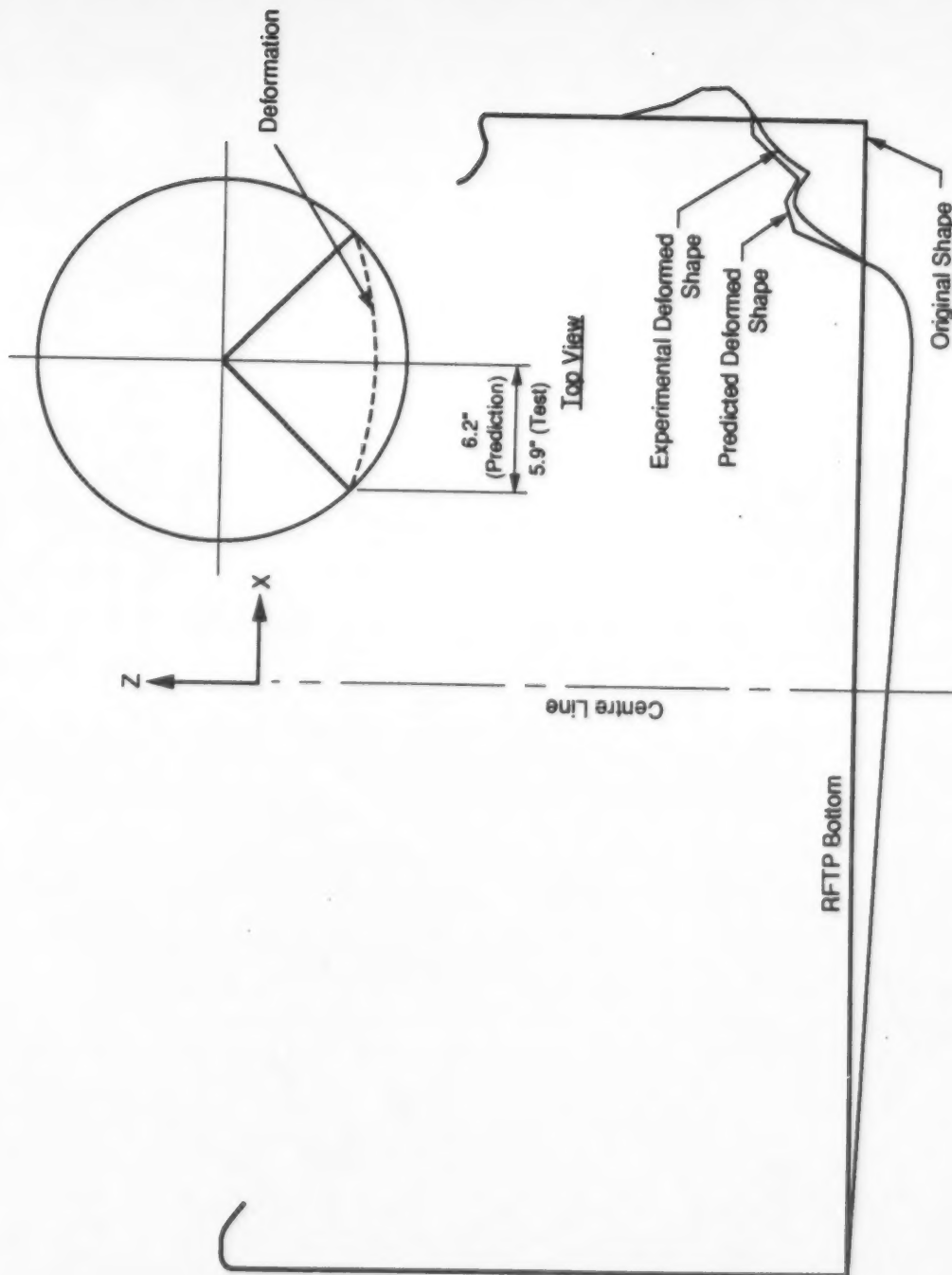


FIGURE 2.69.32

CASE 69: COMPARISON OF PREDICTED AND EXPERIMENTAL RFTP DEFORMATION
AT THE PLANE OF SYMMETRY ($y = 0.0$)

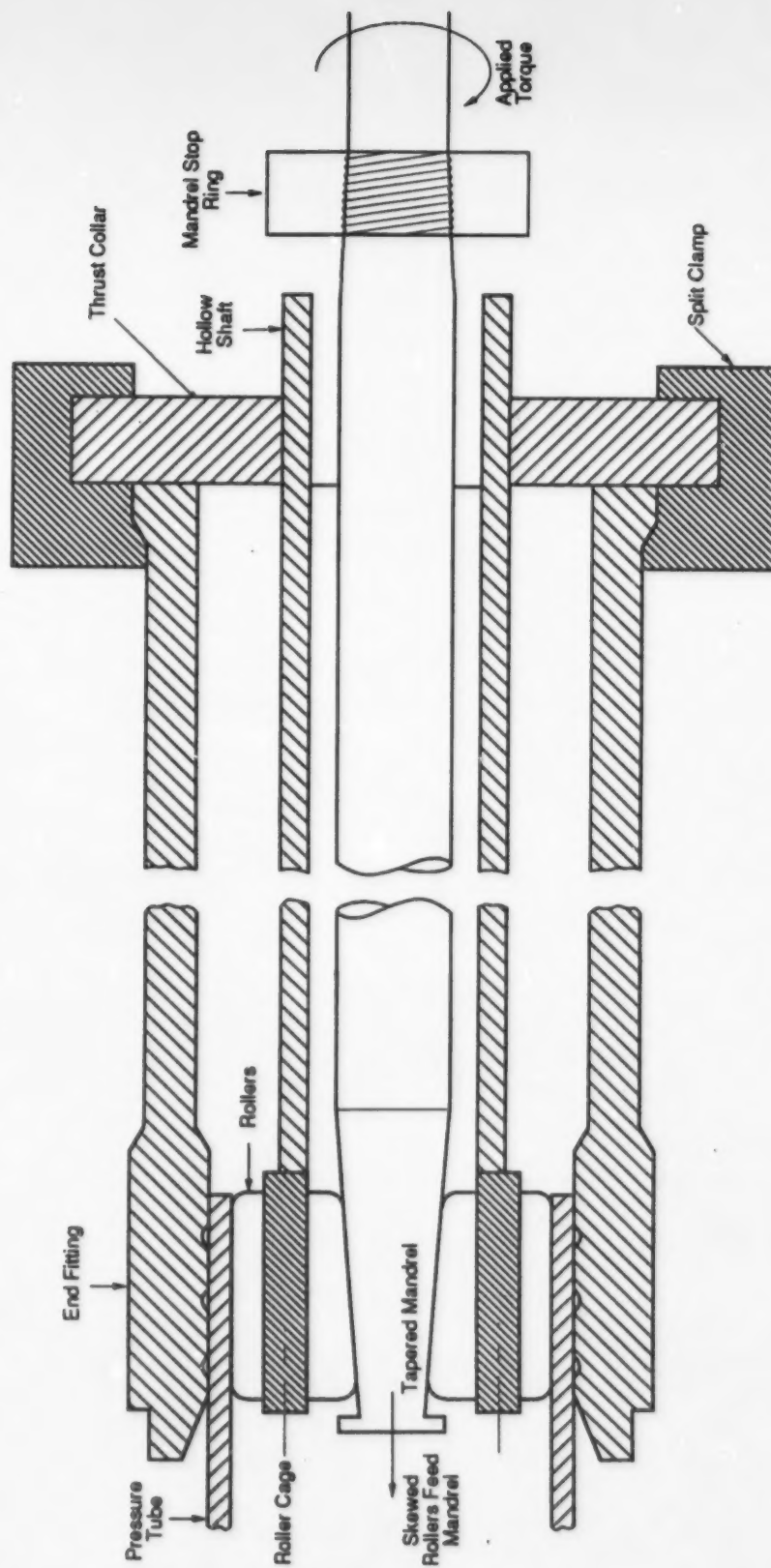
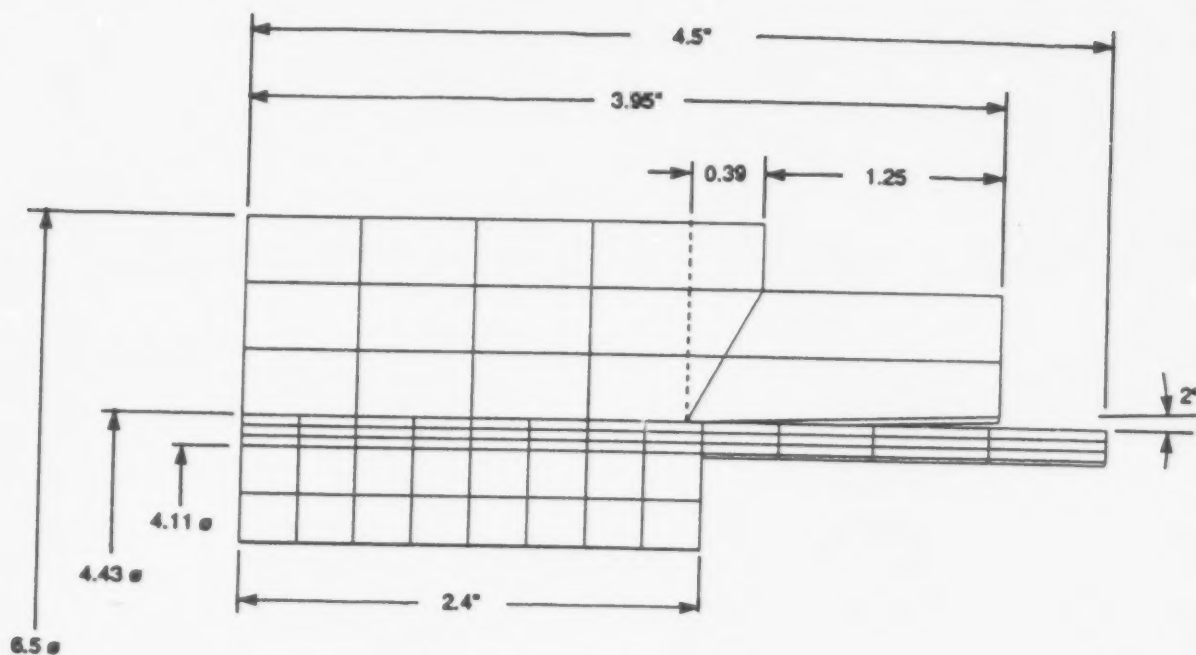
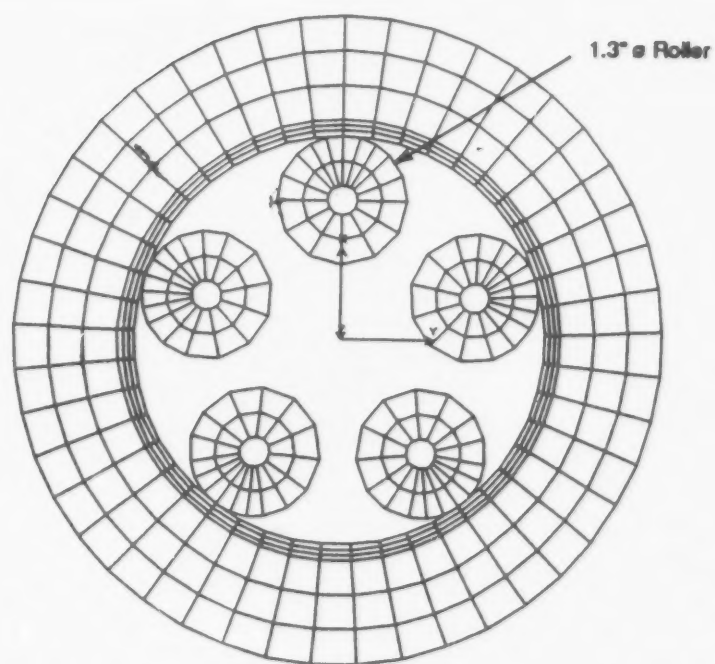


FIGURE 2.70.1
CASE 70: DIAGRAM OF ROLLING PROCESS



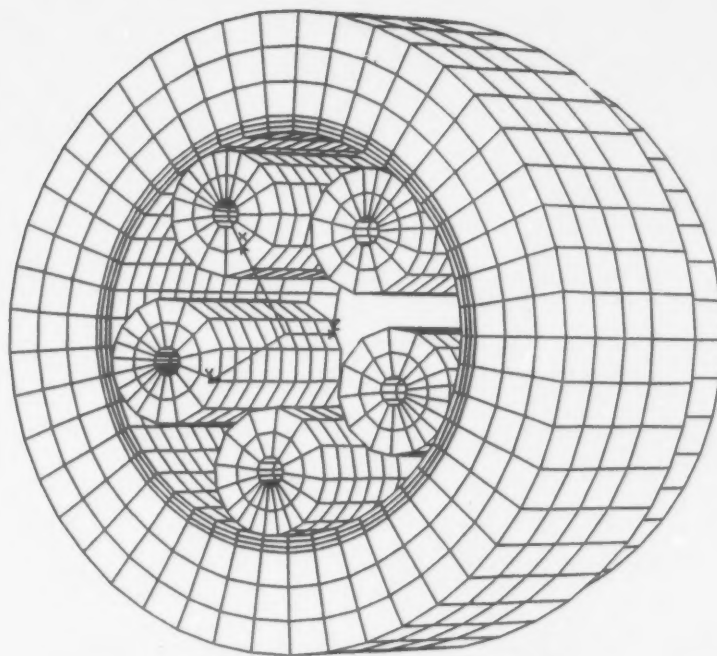
(a) Cross-sectional View of assembly



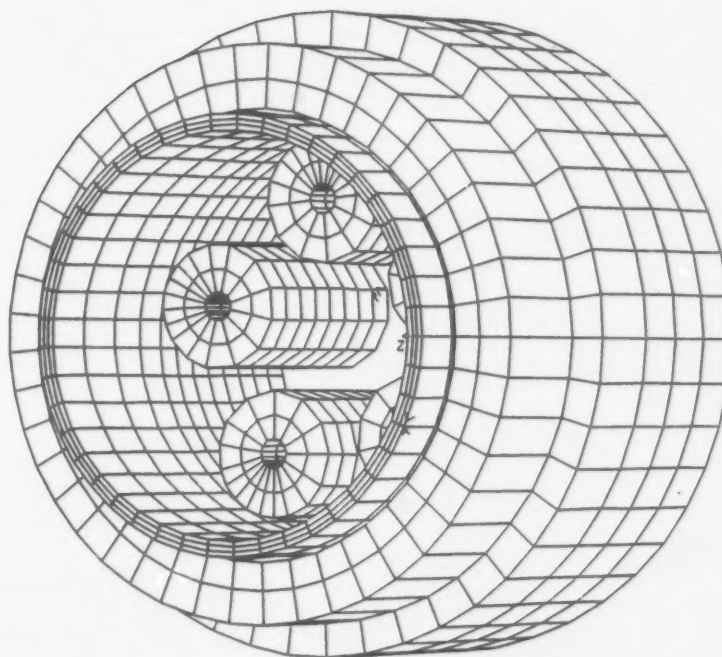
(b) End View of Assembly

FIGURE 2.70.2

CASE 70: ASSEMBLED VIEWS WITH DIMENSIONS



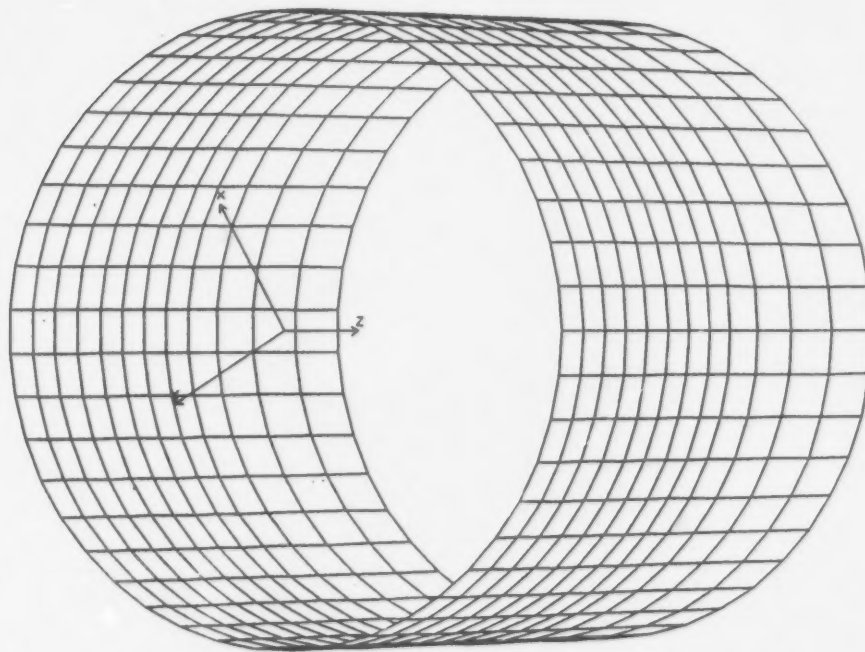
(a)



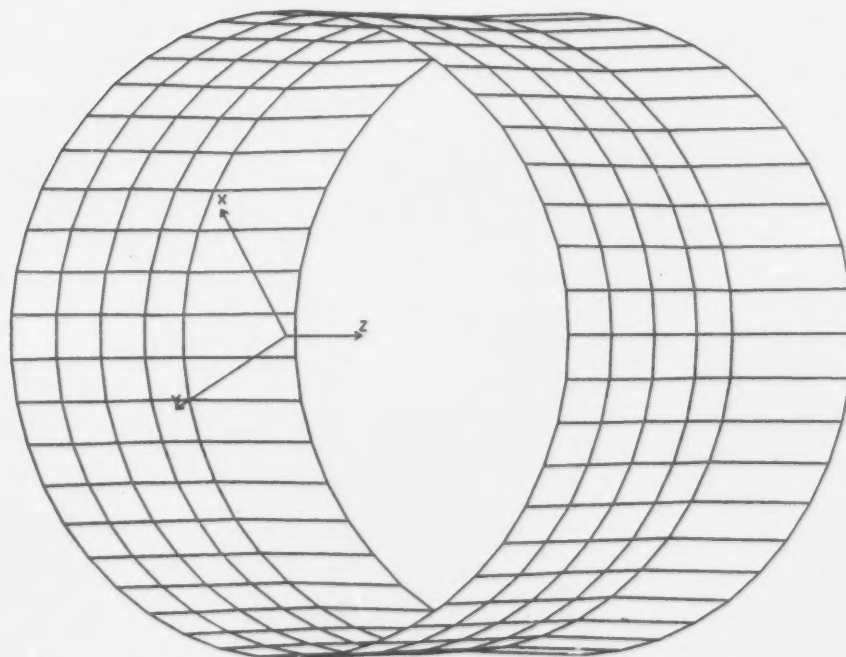
(b)

FIGURE 2.70.3

CASE 70: ASSEMBLED VIEWS OF FINITE ELEMENT MODEL



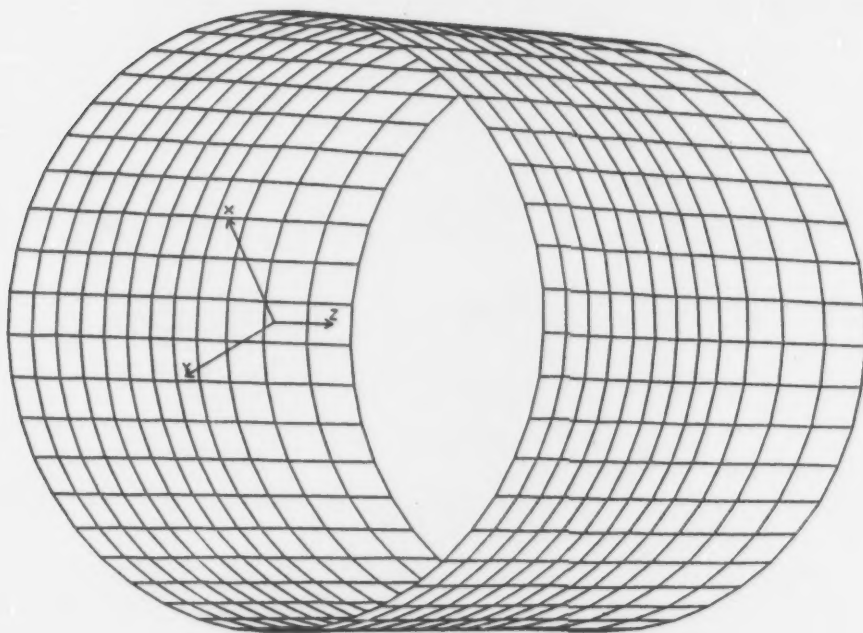
(a) Pressure Tube O.D. Surface



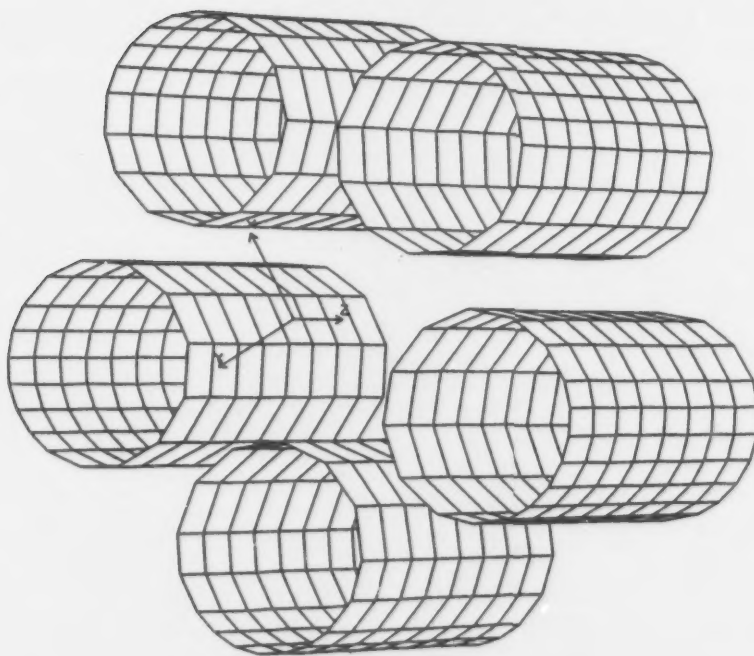
(b) End Fitting I.D. Surface

FIGURE 2.70.4

CASE 70: CONTACT SURFACE SET 1



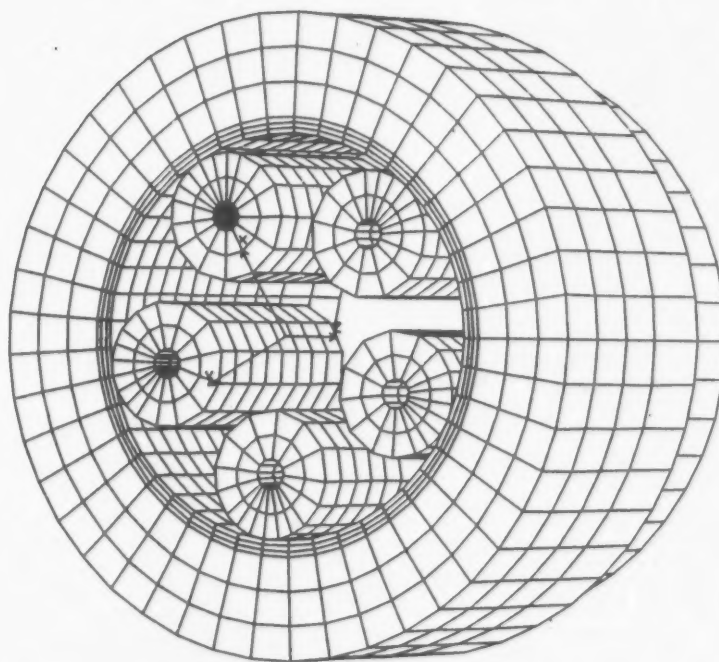
(a) Pressure Tube I.D. Surface



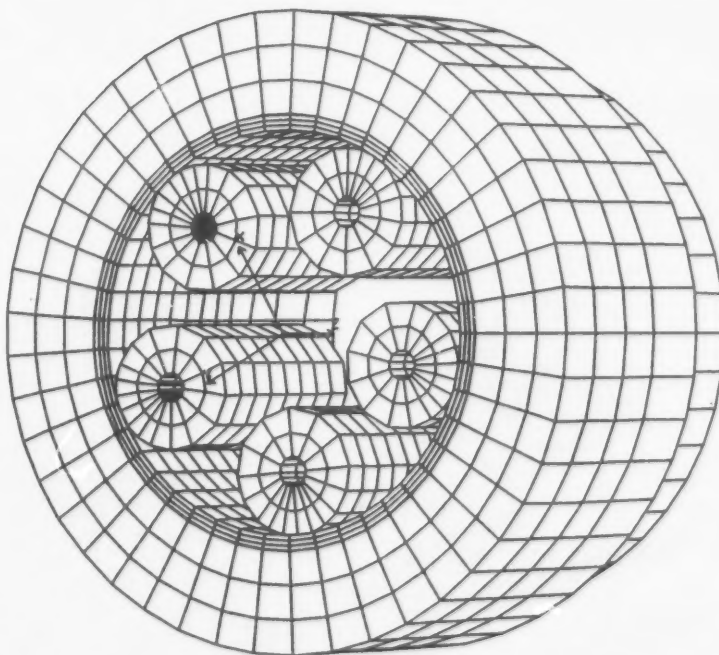
(b) Roller Surfaces

FIGURE 2.70.5

CASE 70: CONTACT SURFACE SET 2



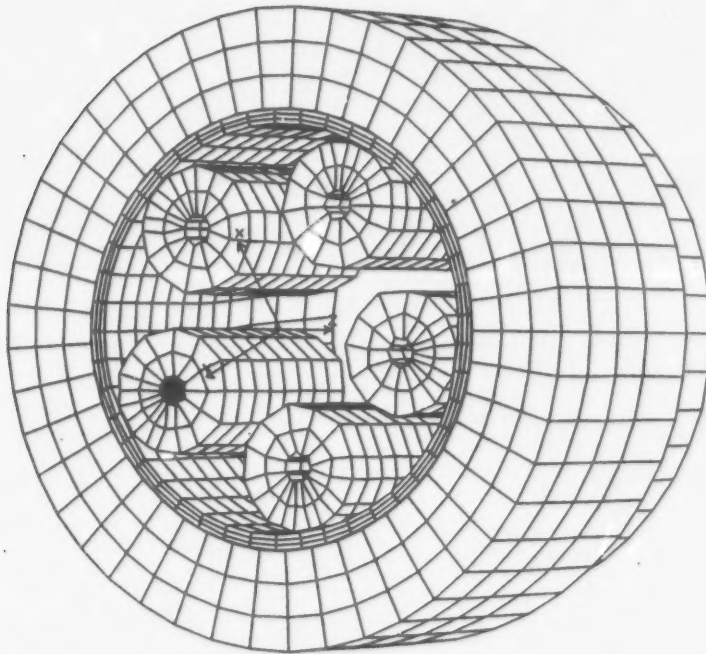
(a) Time = 0.0 ms



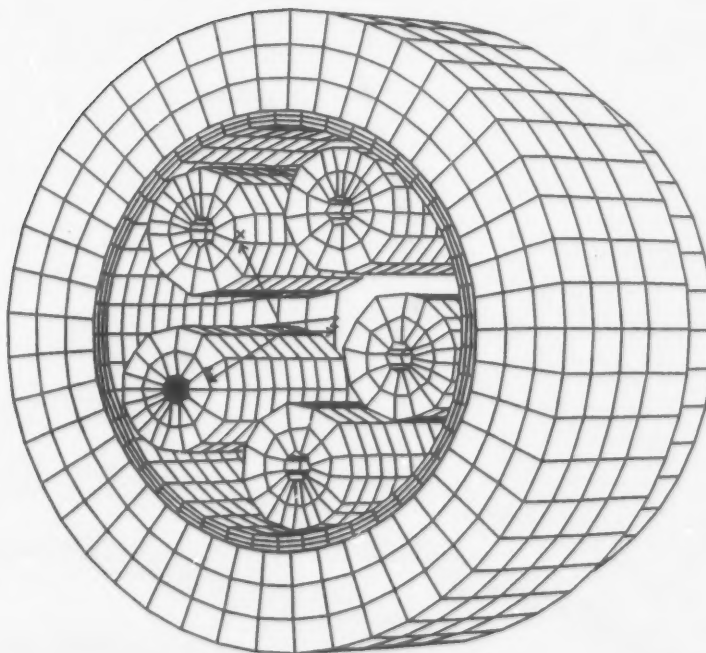
(b) Time = 0.2 ms

FIGURE 2.70.6

CASE 70: PROGRESSION OF ROLLER
MOTION AND DEFORMATION



(c) Time = 2.2 ms



(d) Time = 2.4 ms

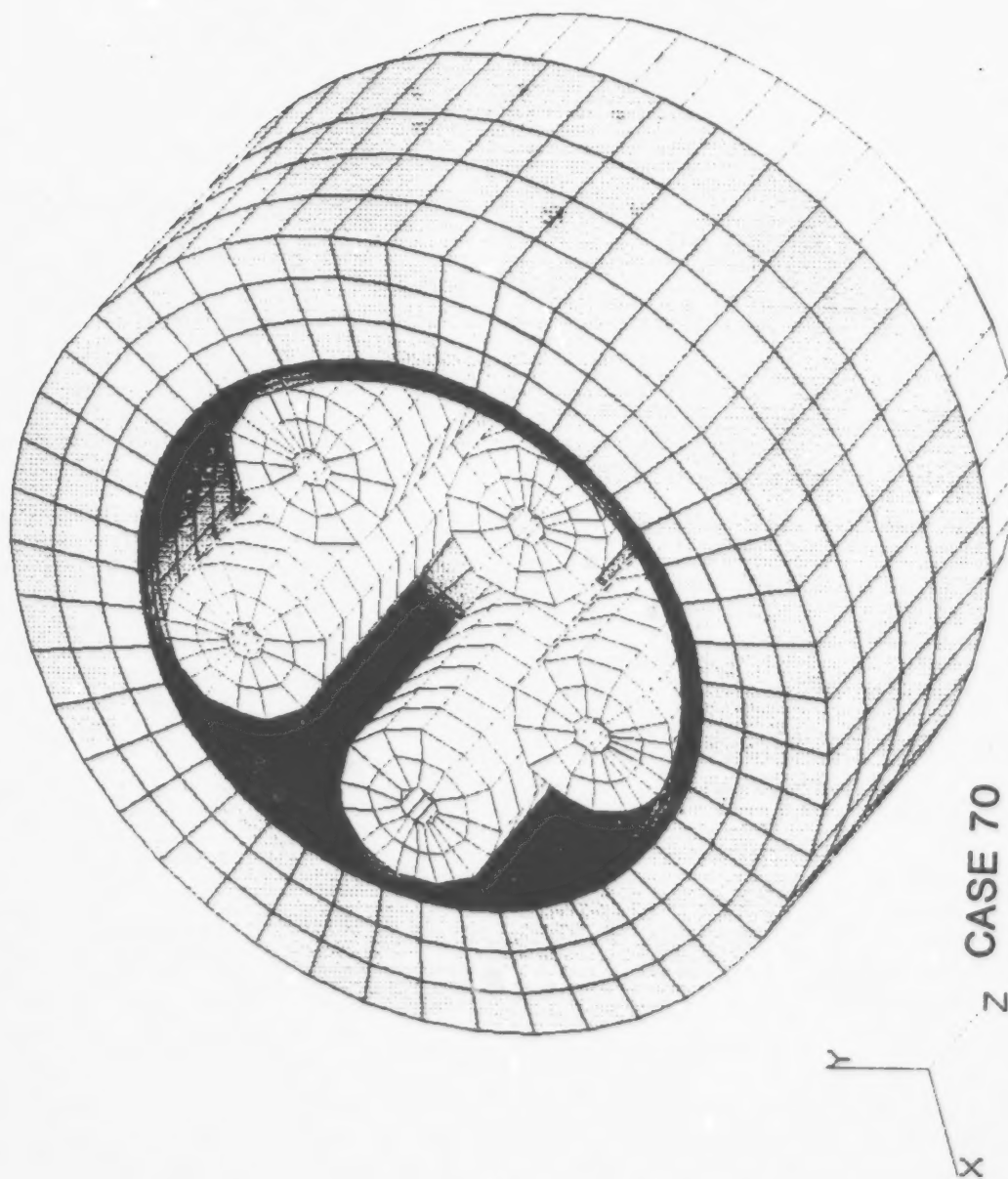
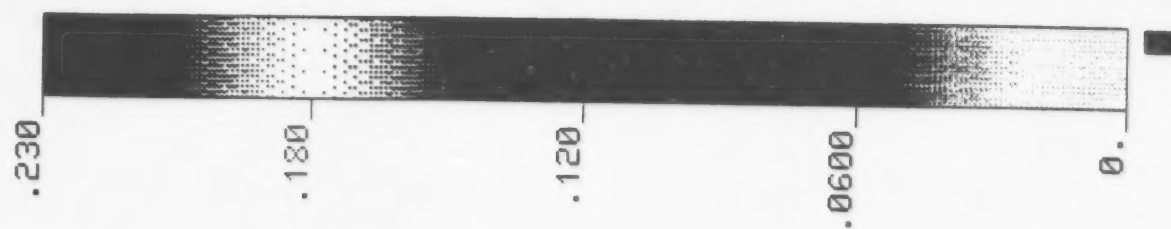
FIGURE 2.70.6

CASE 70: PROGRESSION OF ROLLER
MOTION AND DEFORMATION



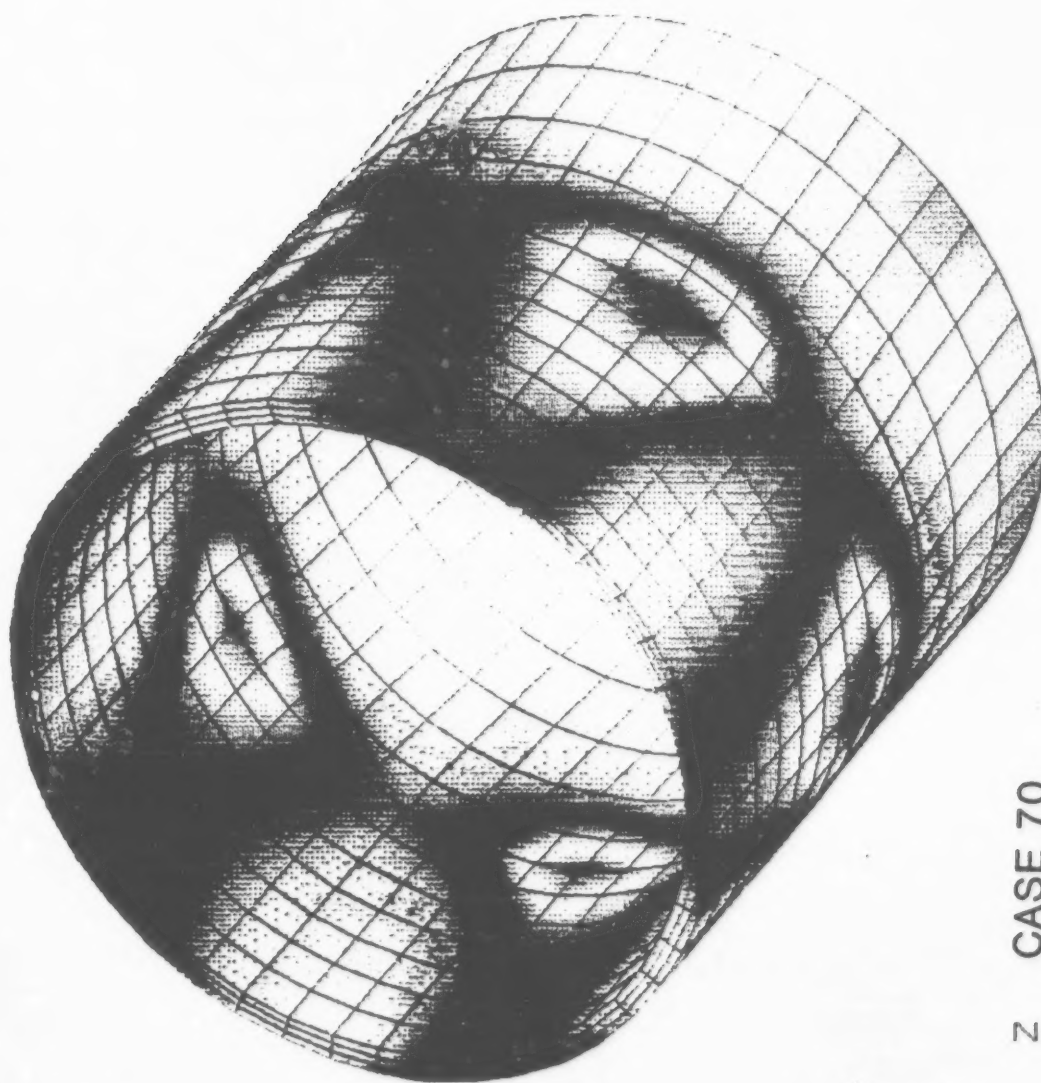
CASE 70

FIGURE 2.70.7 - ASSEMBLED FINITE ELEMENT MODEL
WITH COLOURED COMPONENTS



Z CASE 70

FIGURE 2.70.8 - EFFECTIVE PLASTIC STRAIN
IN ASSEMBLY



Z CASE 70

FIGURE 2.70.9 - EFFECTIVE PLASTIC STRAIN
IN PRESSURE TUBE

3-D CONTACT ALGORITHM TEST CASE 1-D IMPACT -- NODES 1,2,3

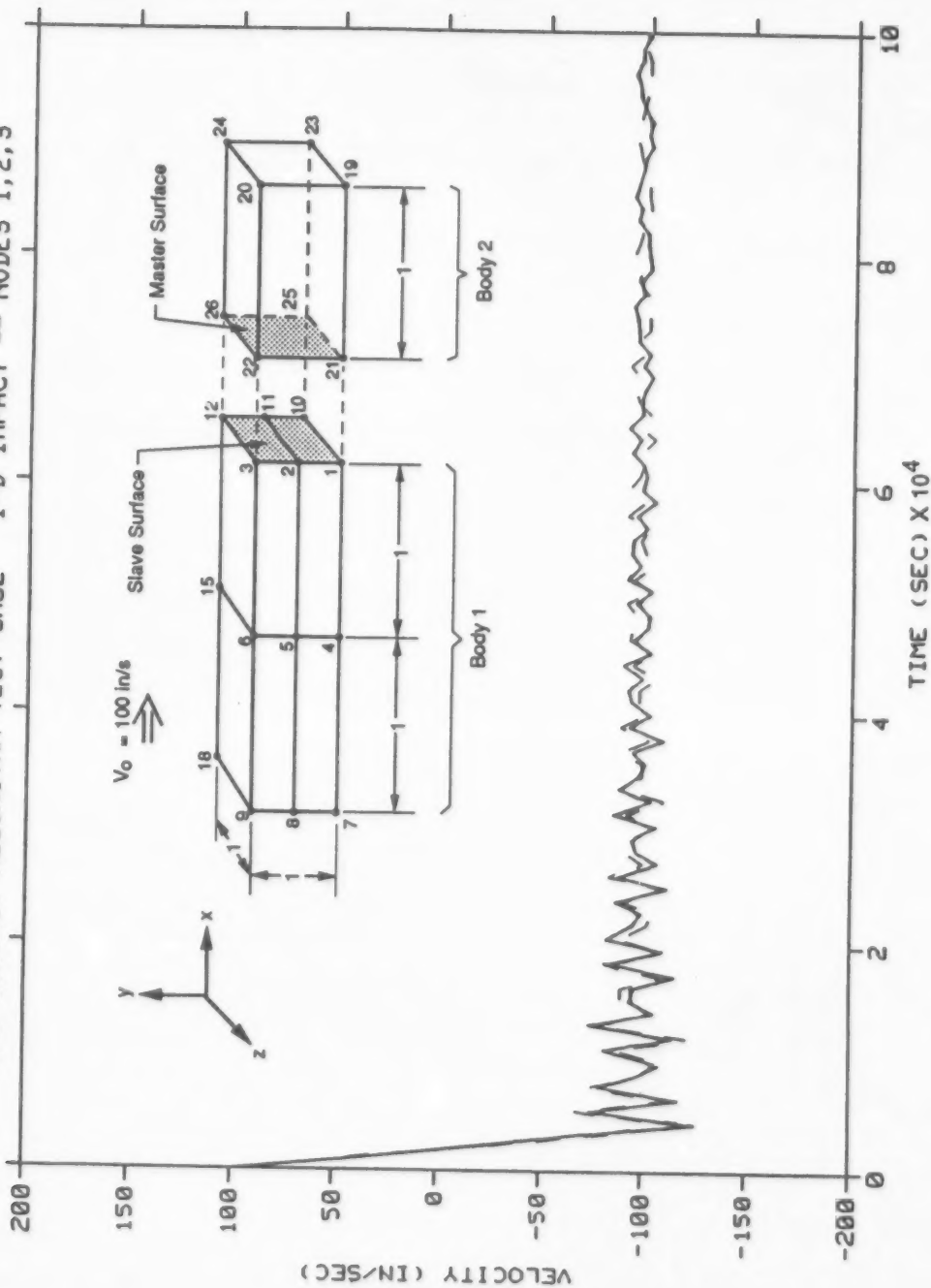


FIGURE 2.71.1
CASE 71: CONTACT ALGORITHM TEST - MOMENTUM TRANSFER

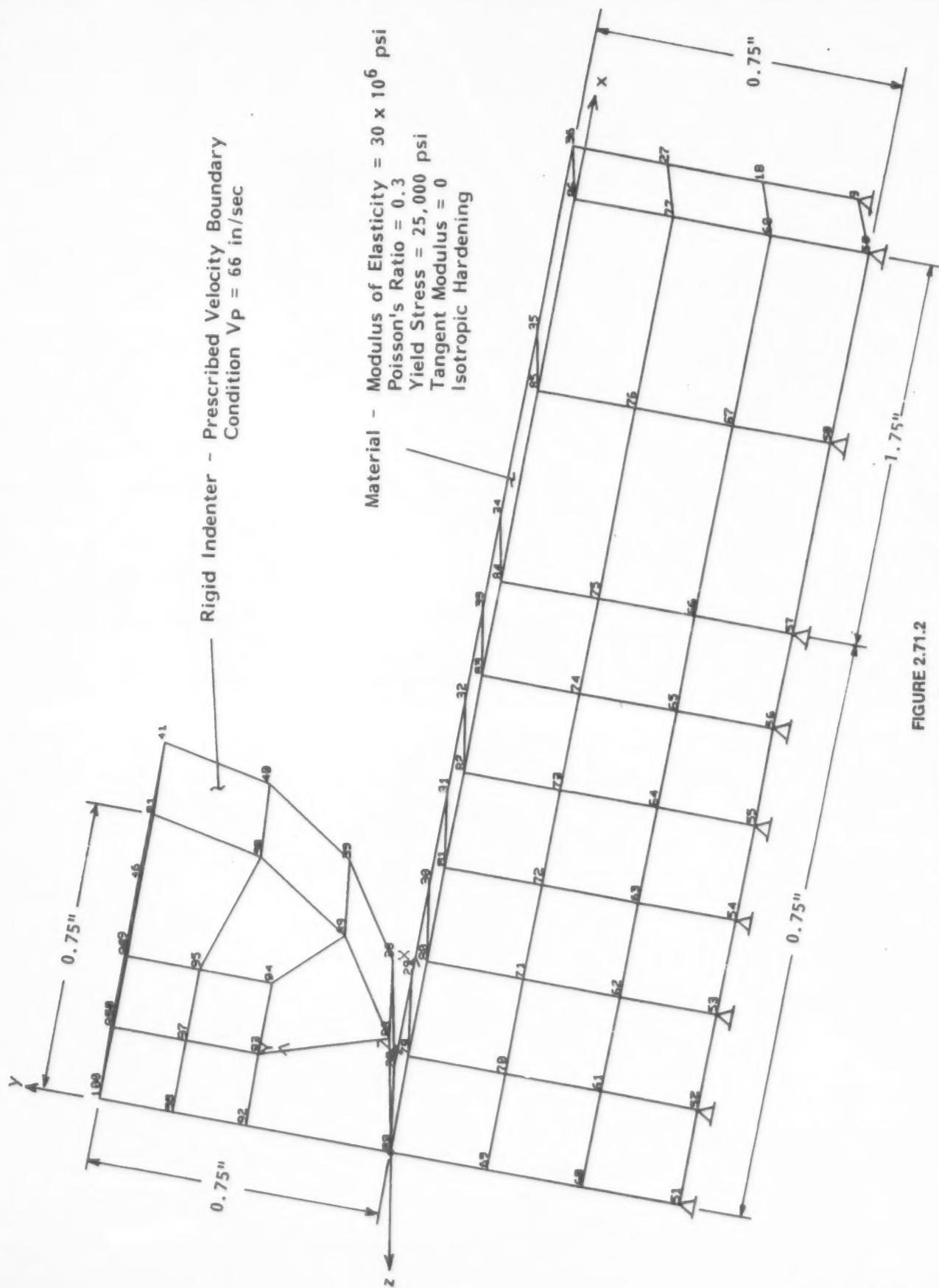


FIGURE 2.71.2

CASE 71: THREE-DIMENSIONAL INDENTOR - MODEL GEOMETRY

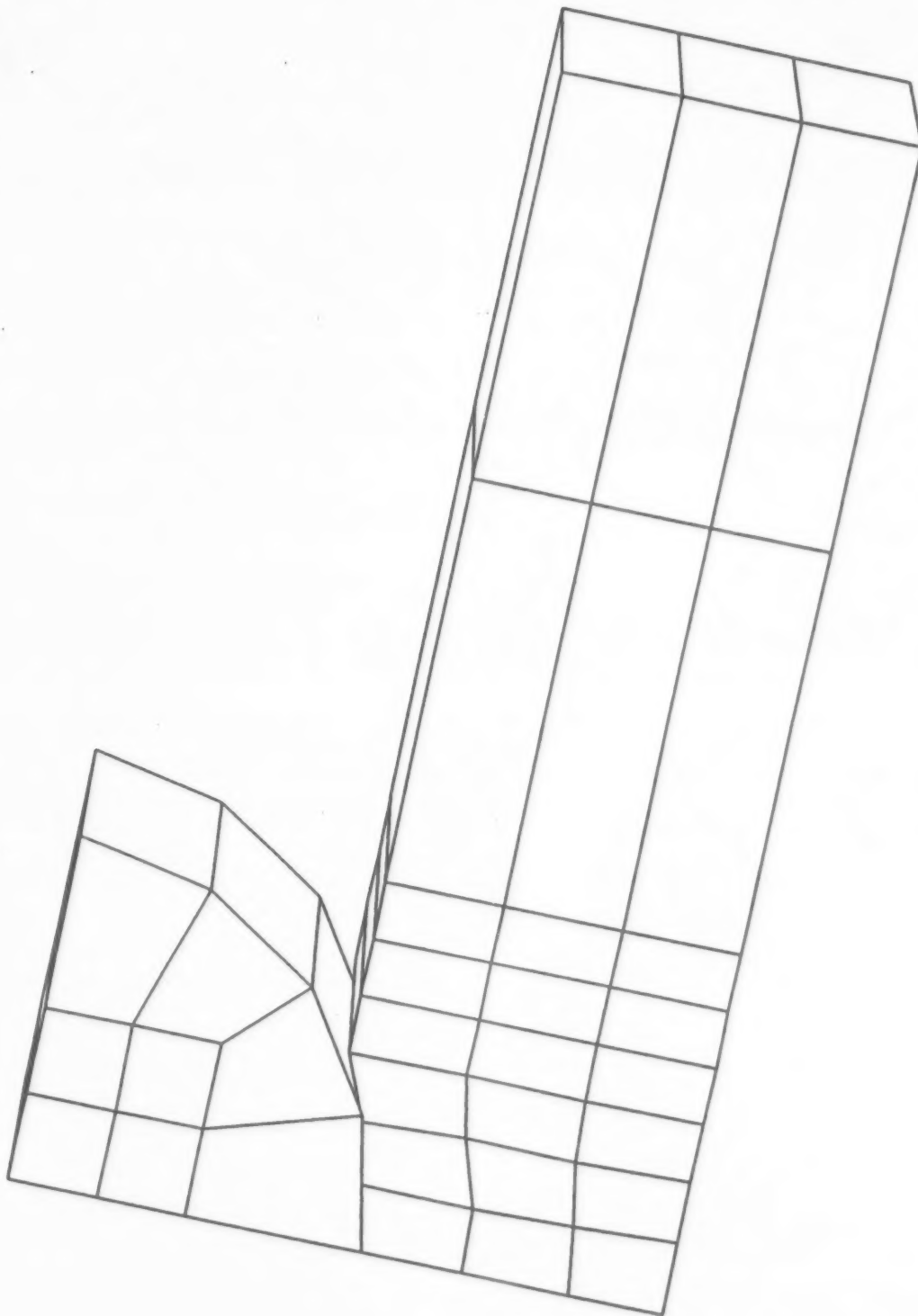


FIGURE 2.71.3

CASE 71: THREE-DIMENSIONAL INDENTOR - DEFORMATION AT TIME ≈ 0.00173 s.

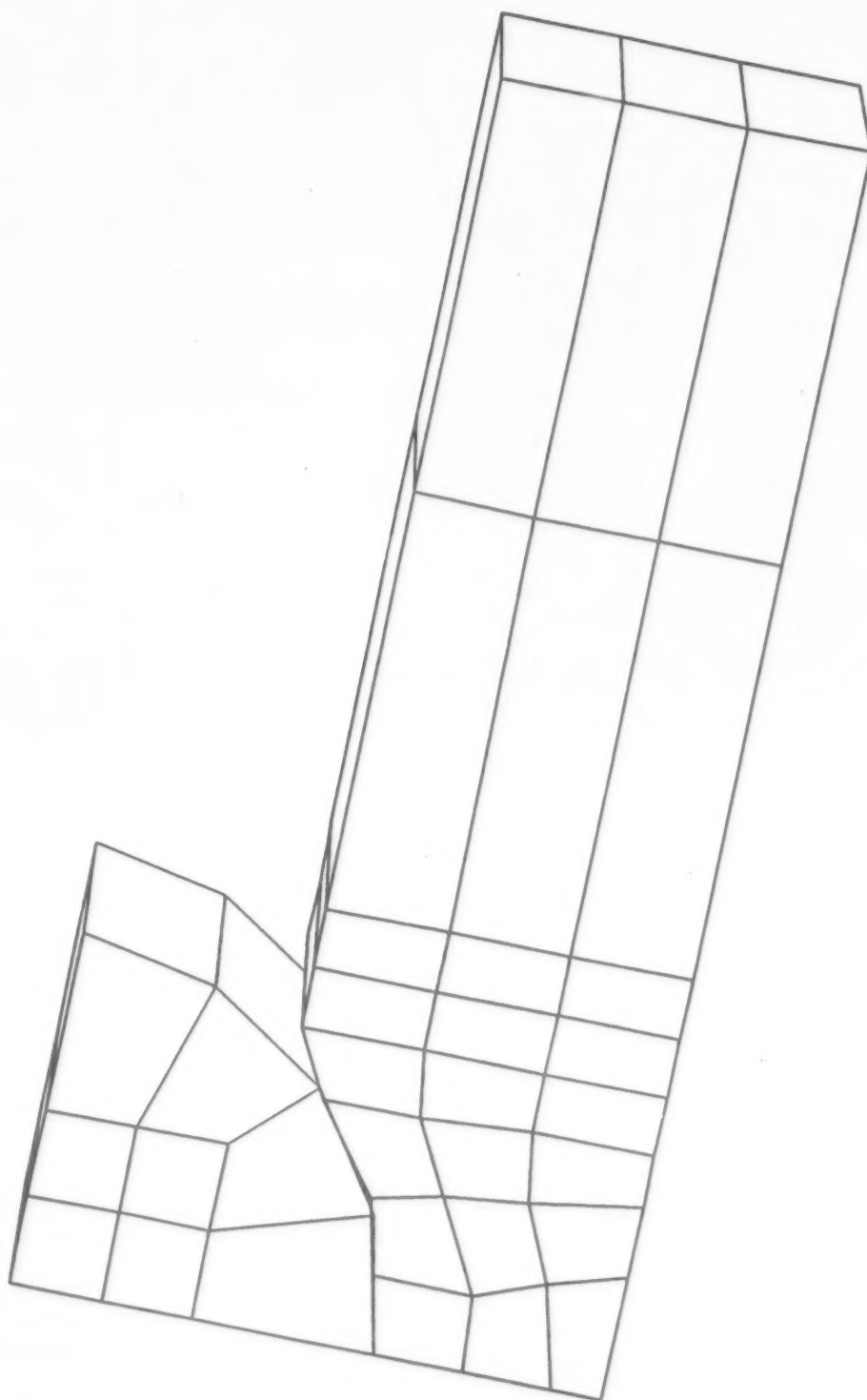
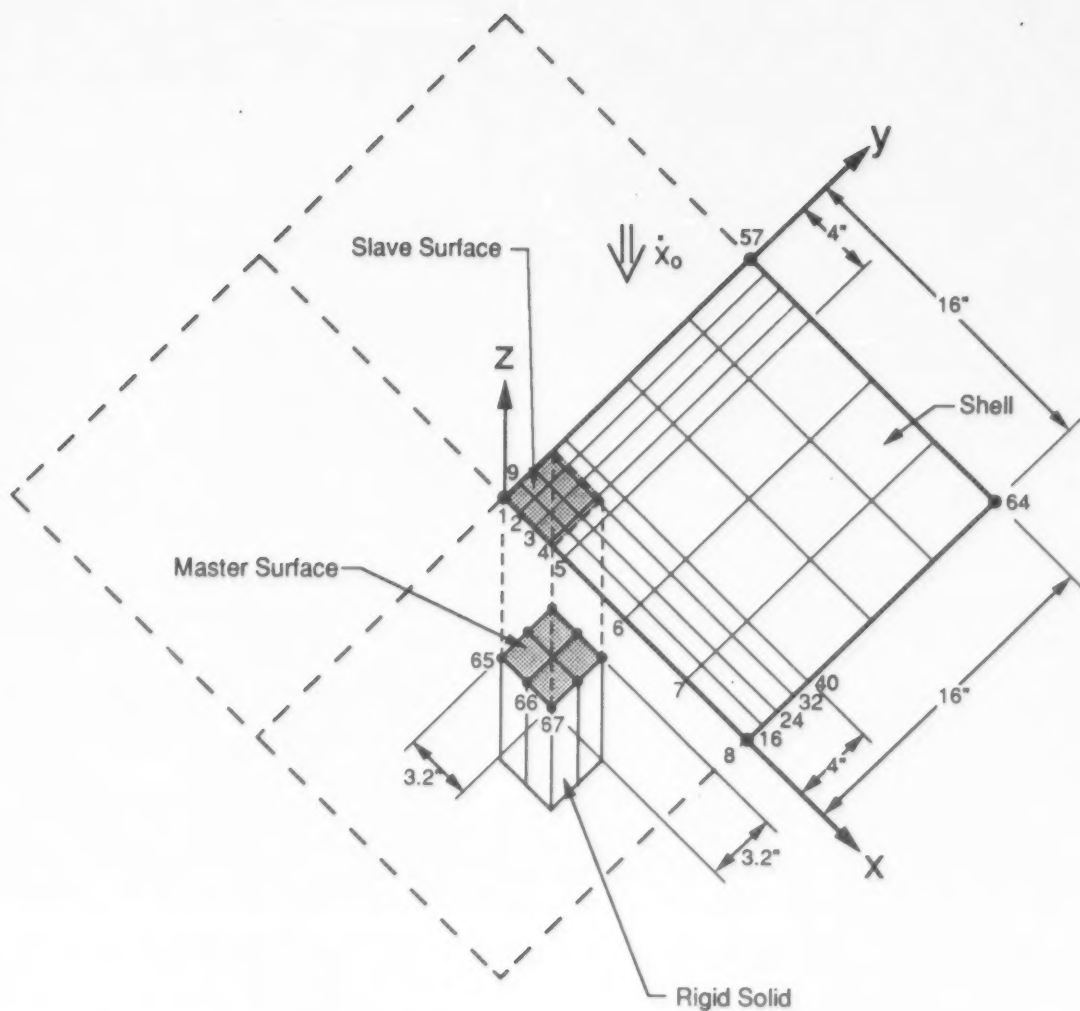


FIGURE 2.71.4

CASE 71: THREE-DIMENSIONAL INDENTOR - DEFORMATION AT TIME = 0.00422 s.



Shell

Thickness = 0.10"
 Modulus of Elasticity = 30×10^6 psi
 Poisson's Ratio = 0.3
 Yield Stress = 20,000 psi
 Tangent Modulus = 0.0

Initial Velocity of Shell = $\dot{X}_0 = -523$ in/s

FIGURE 2.71.5

CASE 71: SHELL-SOLID CONTACT-MODEL GEOMETRY

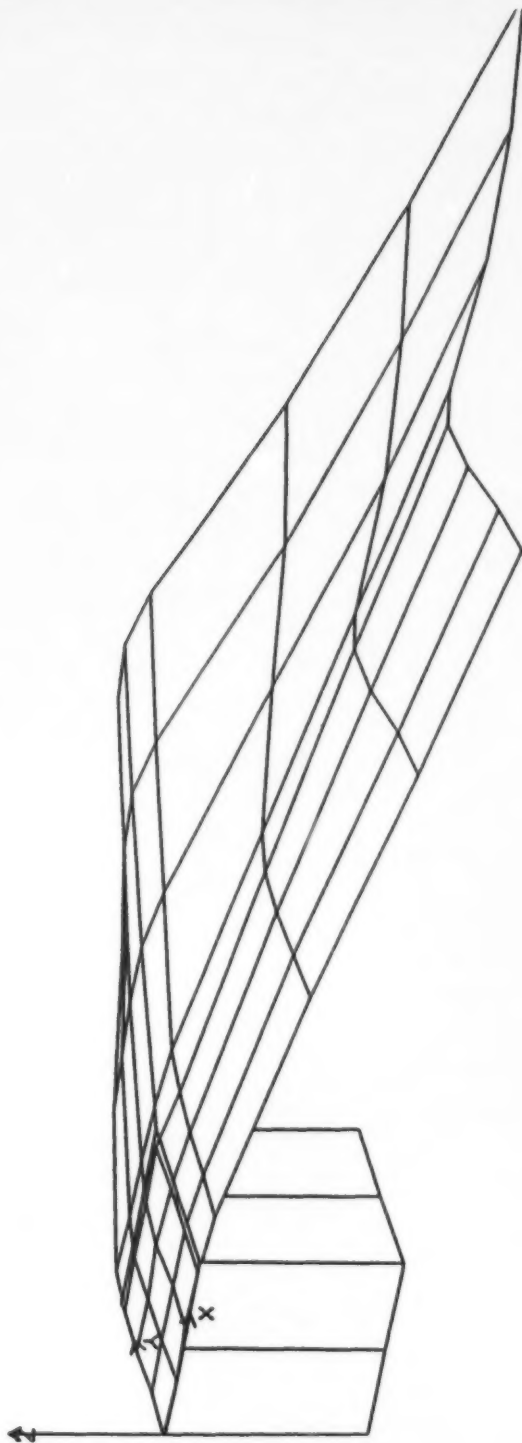


FIGURE 2.71.5

CASE 71: SHELL-SOLID CONTACT - DEFORMATION AT TIME = 0.01 s. (VIEW A)

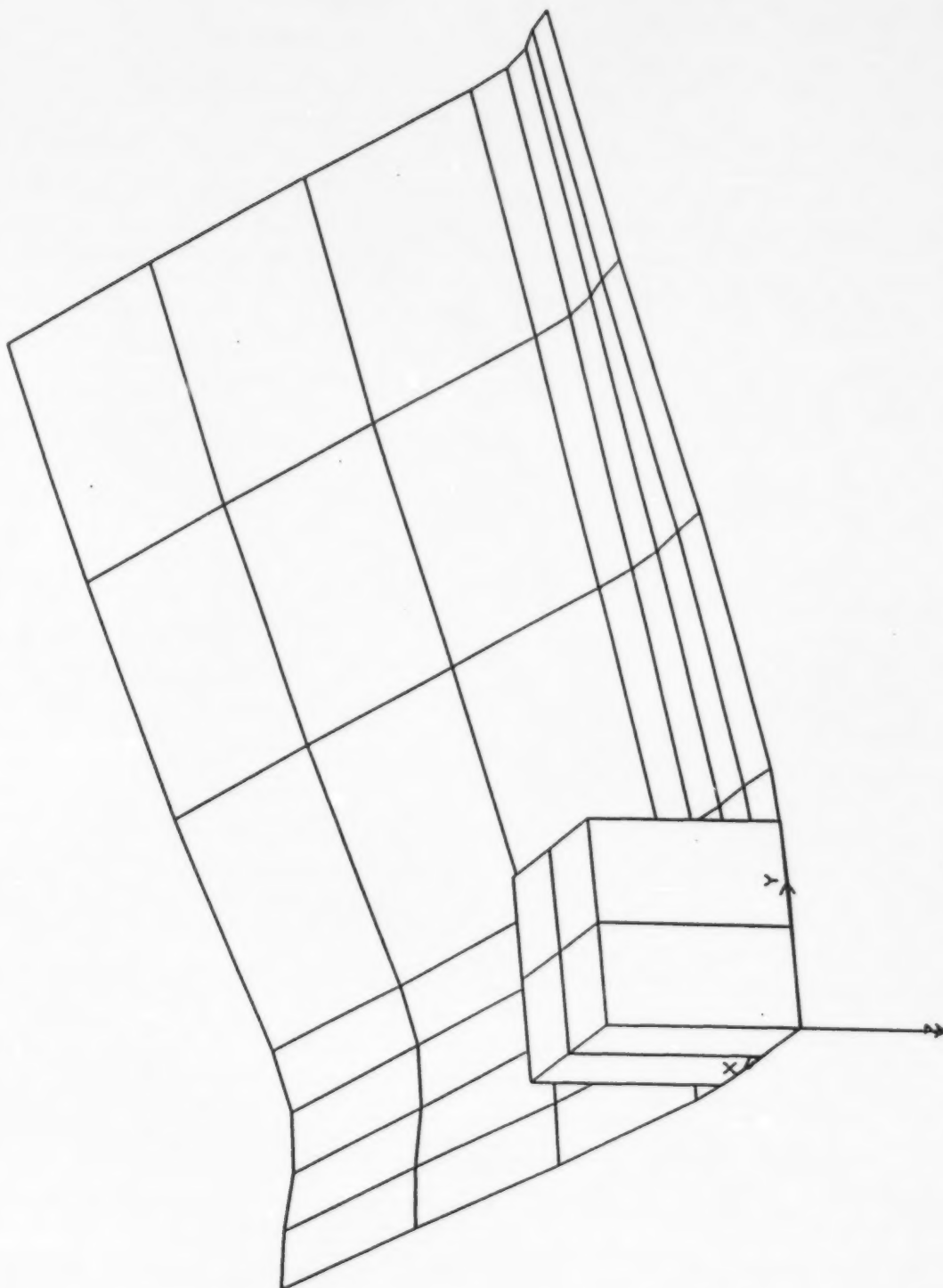


FIGURE 2.71.7

CASE 71: SHELL-SOLID CONTACT - DEFORMATION AT TIME = 0.01 s. (VIEW B)

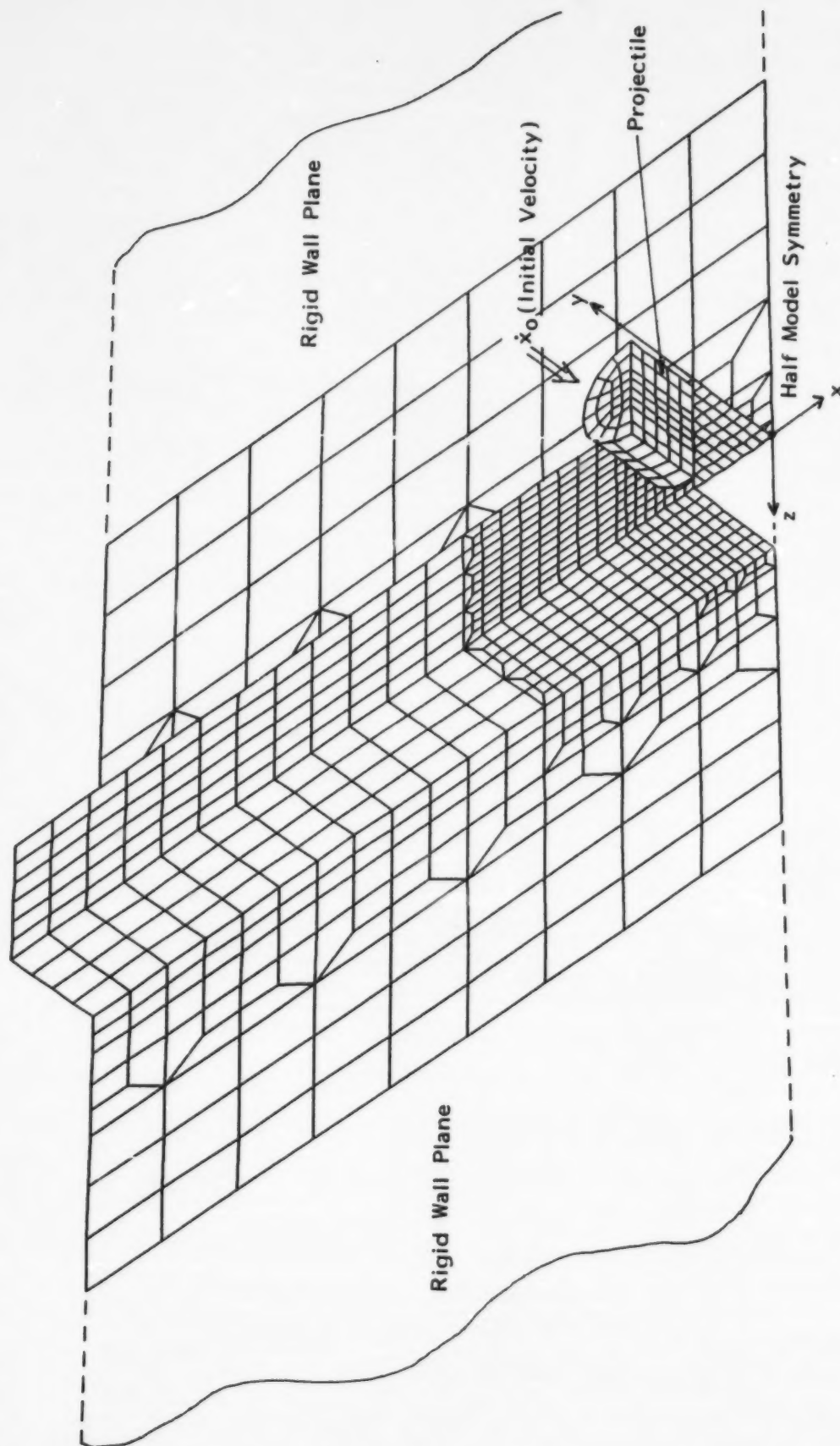


FIGURE 2.71.8

CASE 71: DISCONTINUOUS SURFACE CONTACT MODEL 1 - UNDEFORMED GEOMETRY

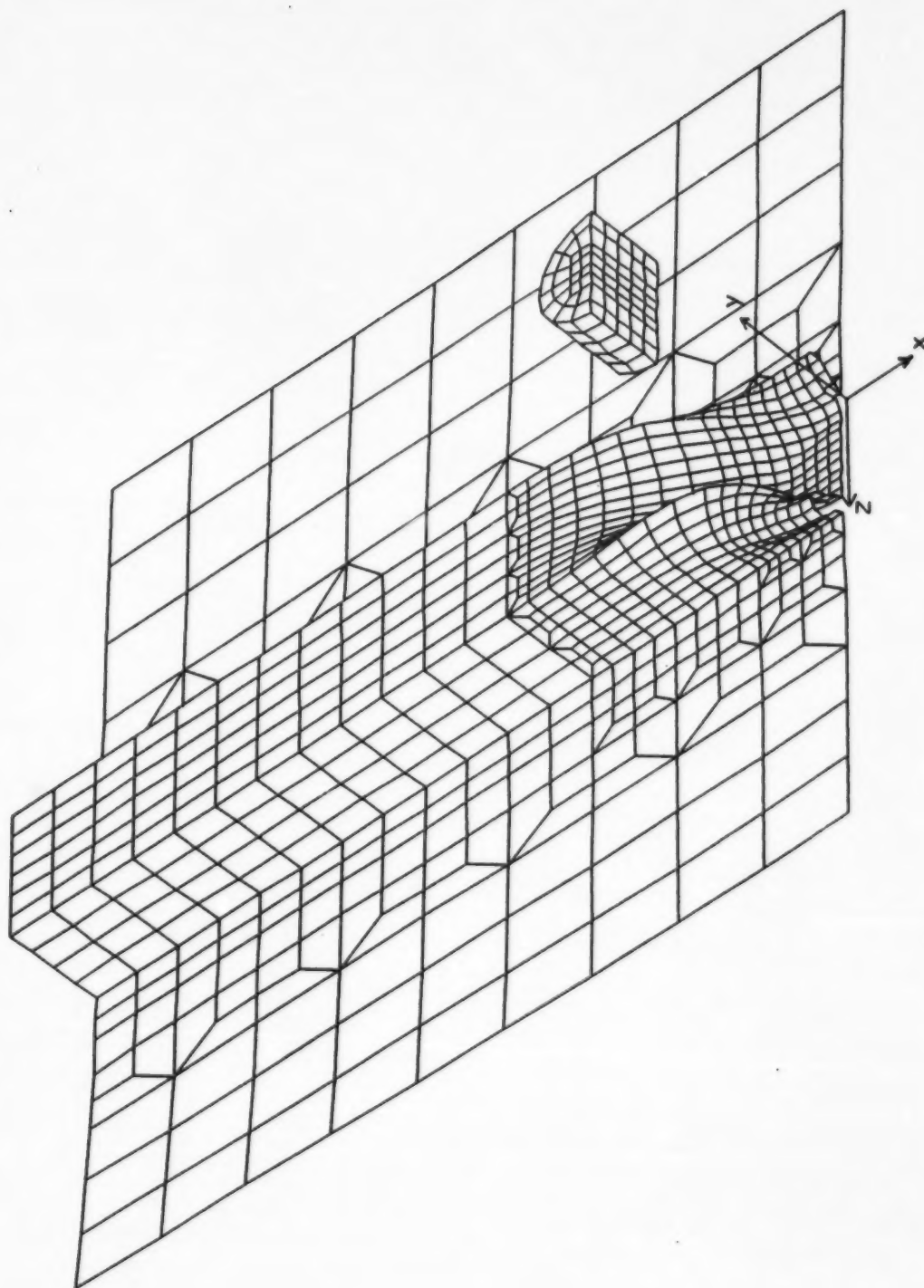


FIGURE 2.71.9

CASE 71: DISCONTINUOUS SURFACE CONTACT MODEL 1 - DEFORMATION AT TIME = 0.007 s. (VIEW A)

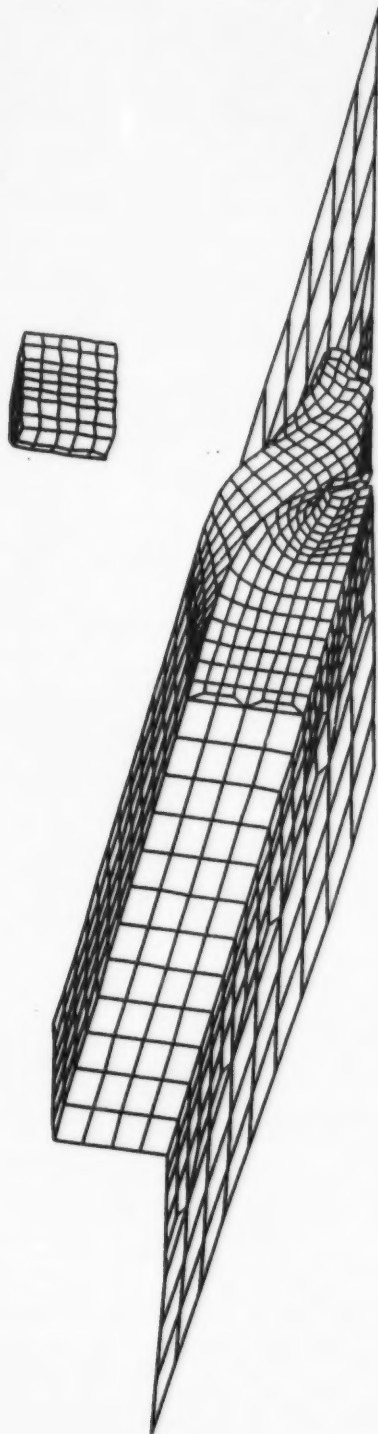


FIGURE 2.71.10

CASE 71: DISCONTINUOUS SURFACE CONTACT MODEL 1 - DEFORMATION AT TIME = 0.007 s. (VIEW B)

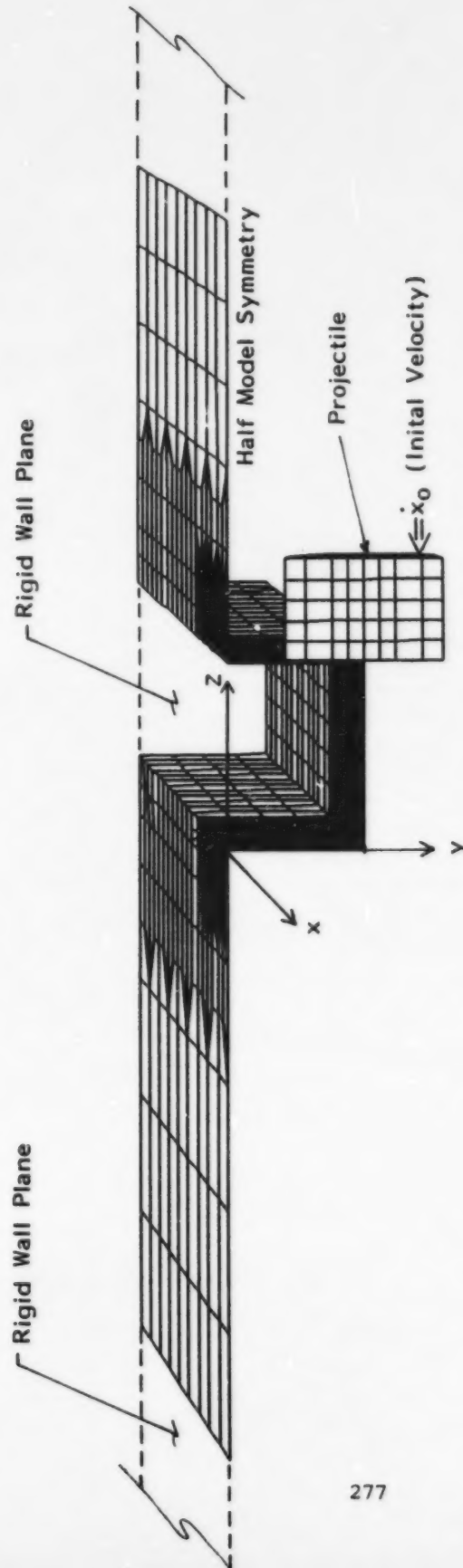


FIGURE 2.71.11

CASE 71: DISCONTINUOUS SURFACE CONTACT MODEL 2 - UNDEFORMED GEOMETRY

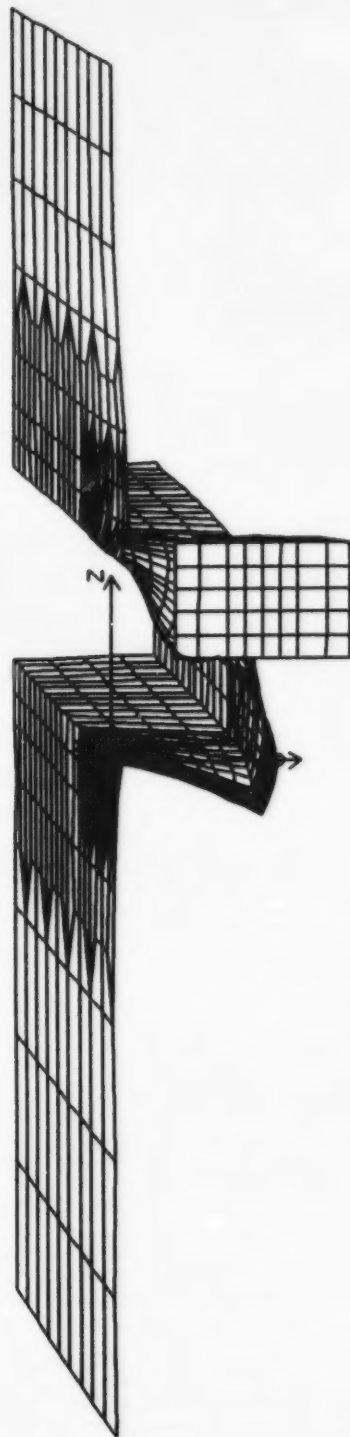


FIGURE 2.71.12

CASE 71: DISCONTINUOUS SURFACE CONTACT MODEL 2 - DEFORMATION AT TIME = 0.001 s.

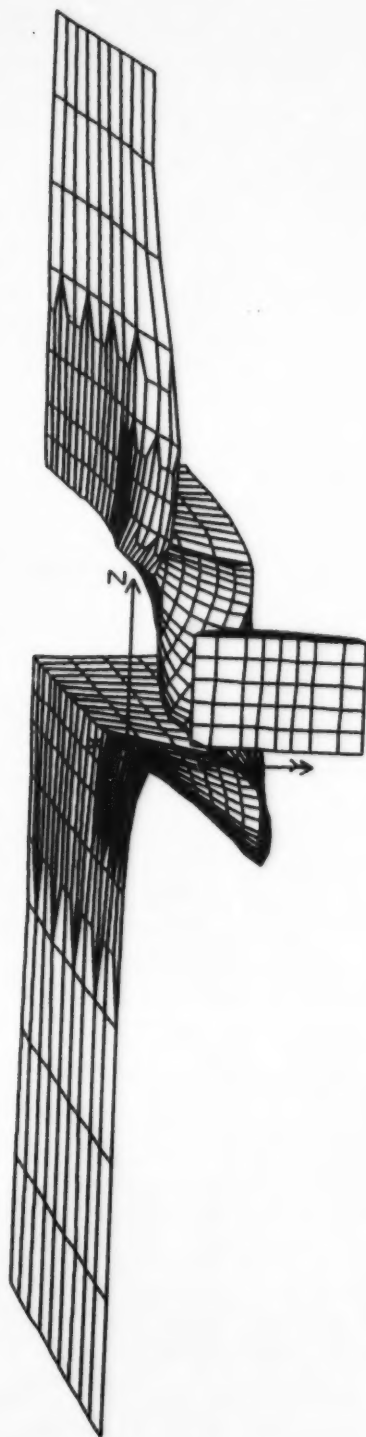


FIGURE 2.71 13

CASE 71: DISCONTINUOUS SURFACE CONTACT MODEL 2 - DEFORMATION AT TIME = 0.002 s.

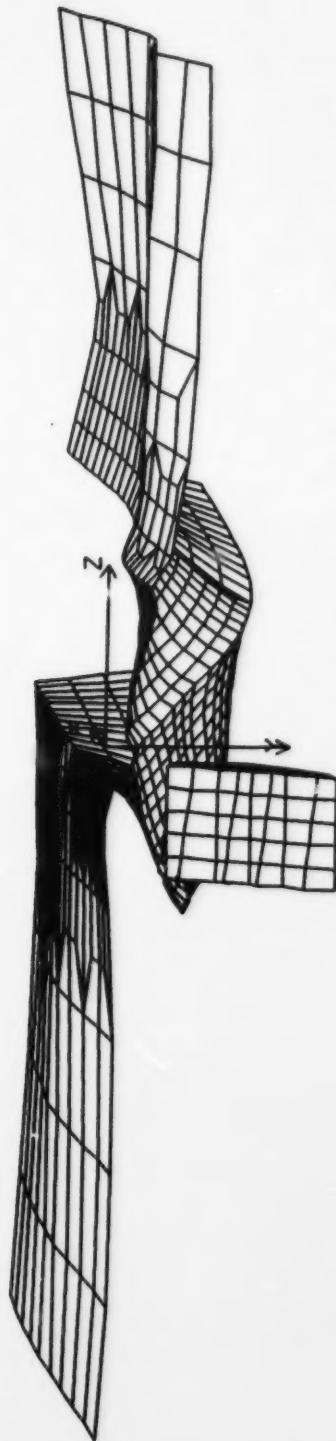


FIGURE 2.71.14

CASE 71: DISCONTINUOUS SURFACE CONTACT MODEL 2 - DEFORMATION AT TIME $\tau = 0.006$ s.

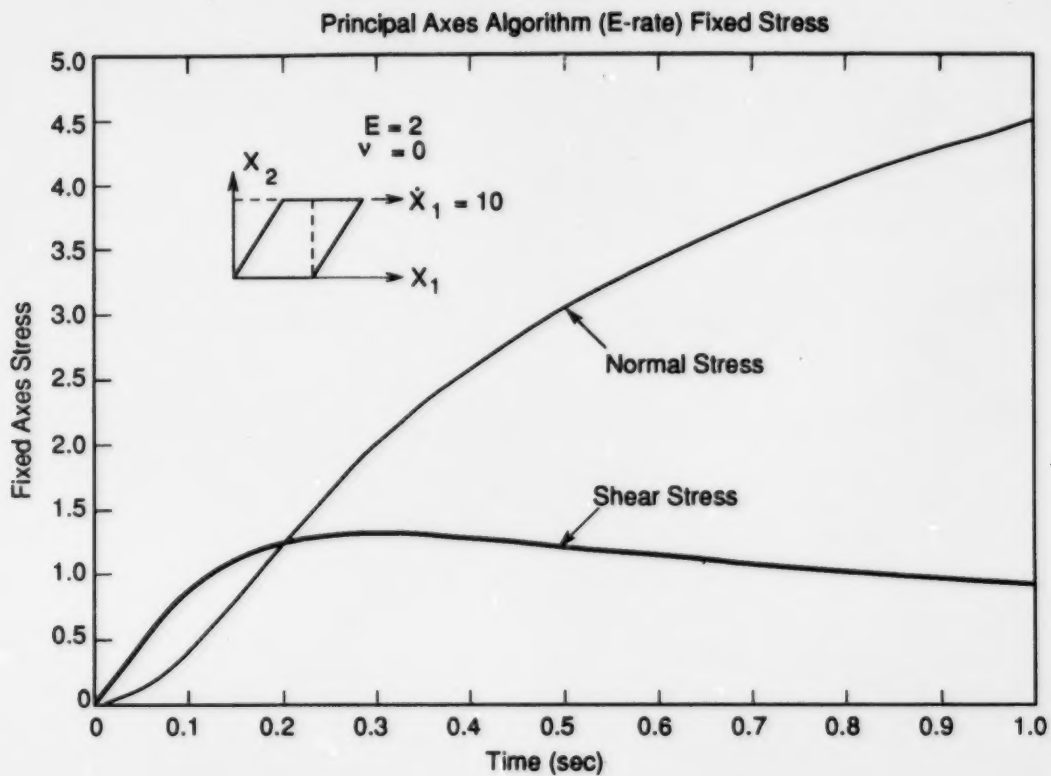


FIGURE 2.72.1

CASE 72: FINITE DEFORMATION SIMPLE SHEAR TEST
USING NUMERICAL ALGORITHM